AutoCAD Civil 2010

# User's Guide

Autodesk

April 2009

# Contents

# Getting Started

# New Features

**1**

AutoCAD Civil 2010 contains many new features and enhancements.

For more information about the new features included in AutoCAD Civil 2010, see the *New Features Workshop*.

## New User Interface Features

**New User Interface Features**

| Feature | Description | Location |
|---------|-------------|----------|
| Ribbon | The ribbon provides a new access method for AutoCAD Civil commands. Displayed at the top of the drawing window, the ribbon provides one location for commands, in an organization that provides the most-frequently used commands in the most accessible places.<br>Object-specific ribbon tabs are displayed when you select objects in the drawing, or they can be opened independently of object selection.<br>For more information, see AutoCAD Civil Ribbon Overview (page 71). | RIBBON command |
| Application Menu | Provides drawing-related commands, such as New, Open, Save, and Export To AutoCAD.<br>The Application Menu is accessible from the button at the upper-left corner of the application window.<br>For more information, see Menus (page 75). | |
| Proximity and rollover tooltips | Provides proximity and roll-over tooltips in an easy-to-read format. For more information, see Object Tooltips (page 106). | Move your cursor over or near objects in the drawing |
| Toolspace tooltips | Provides information about items and collections on the Toolspace tabs. | Move your cursor over objects or collections in the Toolspace |
| Ribbon tooltips | Provides tooltips for ribbon controls, with links to Help for more information. | Move your cursor over ribbon controls. Press F1 to link to a Help topic. |

| Feature | Description | Location |
| --- | --- | --- |
| Object properties exposed in Properties palette | Exposes civil-specific properties in the AutoCAD Properties palette for most civil objects, using a standard format.<br>For more information, see Object Properties (page 57). | Ribbon: View tab ➤ Palettes panel ➤ Properties  |
| Quick Properties | Provides the Quick Properties palette for civil objects.<br>For more information, see Object Properties (page 57). | QPMODE variable |
| Match Properties | Provides support for the Match Properties command for civil objects.<br>For more information, see Object Properties (page 57). | Ribbon: Home tab ➤ Modify panel ➤ Match Properties  |
| Object selection from Prospector | Provides new commands for selecting objects in the drawing from the Toolspace Prospector tab. | Toolspace Prospector tab ➤ right-click object ➤ Select |
| Data Shortcuts folder in Prospector | The Data Shortcuts folder is now available in the Active Drawing View. For more information, see Using Data Shortcuts (page 127). | Toolspace Prospector tab ➤ select Active Drawing View from the drop-down list |

# New Alignments Features

**New Alignments Features**

| Feature | Description | Location |
| --- | --- | --- |
| Create from objects | The Create Alignment From Polyline command has been renamed to Create Alignment From Objects. The command now provides the ability to select multiple lines, arcs, and polylines to use as the alignment geometry. For more information, see Creating an Alignment from Graphic Entities (page 891). | Ribbon: Home tab ➤ Create Design panel ➤ Alignments ➤ Create Alignment From Objects  |
| New Type property for alignments | Provides a Type property for alignments. Types include centerline, offset, miscellaneous, and curb return.<br>Alignment creation dialog boxes provide a control for specifying alignment type. All existing alignments from previous version drawings are assigned the centerline type.<br>In the Toolspace Prospector tree, alignments are organized according to their type.<br>For more information, see Alignment Types (page 855). | Alignment creation dialog boxes<br>Alignment Properties dialog box |
| New Alignment Properties | Provides a new tab for masking in the Alignment Properties dialog box.<br>The masking feature enables you to define ranges on an alignment where the geometry and labels are not drawn (but still exist). For more information, see To mask an alignment (page 865). | Alignment Properties dialog box ➤ Masking tab |
| Create offset alignments | Creates offset alignments that are dynamic to the parent alignment. For more information, see Creating Offset Alignments (page 895). | Ribbon: Home tab ➤ Create Design panel ➤ Alignment ➤ Create Offset Alignment |

| Feature | Description | Location |
|---------|-------------|----------|
| | |  |
| Create widening | Creates rule-based non-parallel offsets (such as bus bays and turn lanes with widening and transitions) at specified stations or geometry locations. For more information, see Creating Widening (page 896). | Ribbon: Home tab ➤ Create Design panel ➤ Alignment ➤ Create Widening  |
| Make dynamic offsets or curb returns static | Offset alignments are dependent on a parent alignment and curb returns are dependent on two parent alignments. These alignments react dynamically to their parent alignments. However, they can be changed to a static state for manual editing. For more information, see Offset Parameters Tab (Alignment Properties Dialog Box) (page 1757). | Alignment Properties dialog box, Offset Parameters /Curb Returns Parameters tab, Update Mode setting |

# New API Features

**New API Features**

| Feature | Description |
|---------|-------------|
| .NET API for selected civil objects | Provides a .NET API for Alignments, Styles, Settings, Profiles, and Pipe Networks. For more information, see .NET API. |

# New Corridors Features

**New Corridors Features**

| Feature | Description | Location |
|---------|-------------|----------|
| Section Editor | Provides the Corridor Section Editor commands on a ribbon tab. For more information, see Viewing and Editing Corridor Sections (page 1361). | Ribbon: Corridor context tab ➤ Modify panel ➤ Corridor Section Editor  |
| Clearing geometric overrides from corridor sections | Provides a streamlined method of clearing overrides that have been applied to cross sections. A check box has been added to the Assembly level collection in the Parameter Editor of the View/Edit Corridor Section tools. This check box is selected if overrides have been applied at a lower level. You can clear this check box to clear all the overrides that are applied at the lower level. For more information, see To override subassembly parameters by editing their values (page 1365). | Section Editor tab ➤ Corridor Edit Tools panel ➤ Parameter Editor. Override check box available at the Assembly collection level |
| Select region button | Provides a button on the Parameters tab of the Corridor Properties dialog box to select a region to make it current in the drawing. For more information, see Parameters Tab (Corridor Properties Dialog Box) (page 1815). | Corridor Properties dialog box, Parameters tab, Select Region From Drawing button |
| Naming template for baselines and regions | Provides a naming template for new corridor baselines and new/inserted corridor regions. | Corridor Properties dialog box, Parameters tab. |

| Feature | Description | Location |
|---|---|---|
| | | When you insert a new baseline or region, you can access the Name Template dialog box. |
| Edit Region | Provides an Edit Region command, which opens the Corridor Properties dialog box and makes the selected region current in the dialog box. For more information, see Adding and Editing Corridor Regions (page 1337). | Ribbon: Corridor context tab ➤ Modify panel ➤ Edit Region |
| Isolate/Show Region | Provides an Isolate Region command that turns off all other regions in the corridor, so that only the desired region is visible on the screen. Provides a Show All Regions command to turn on the visibility of any regions that are currently turned off. For more information, see Adding and Editing Corridor Regions (page 1337). | Ribbon: Corridor context tab ➤ Modify panel ➤ Isolate Region Ribbon: Corridor context tab ➤ Modify panel ➤ Show All Regions |
| Persistent targets | Provides persistent targets when changing assemblies on the Corridor Properties dialog box Parameters tab (for common target types). For more information, see To edit a corridor region (page 1337). | Corridor Properties dialog box, Parameters tab |
| Highlighting | Provides new Region Highlight Settings to control the highlighting of regions and baselines in the corridor when editing the corridor parameters in the Corridor Properties dialog box. For more information, see To change corridor creation settings (page 1334). | Toolspace Settings tab ➤ right-click Corridor collection ➤ Edit Feature Settings |
| Zoom to and Pan to region | Provides new commands to zoom and pan to corridor regions from the Corridor Properties dialog box. For more information, see Parameters Tab (Corridor Properties Dialog Box) (page 1815). | Corridor Properties dialog box, Parameters tab |
| Automatic boundary creation for corridors with multiple baselines | Provides a new automated method of creating boundaries for corridors with multiple baselines, creating a shrink wrap boundary around the entire corridor. This aids in surface creation for the corridor, as well as representing excavation limits. For more information, see To automatically create outer boundaries (page 1356). | Corridor Properties dialog box, Boundaries tab. In the dialog box, right-click corridor name ➤ Corridor Extents As Outer Boundary. NOTE This command is shown in the menu only if the corridor has multiple baselines. |
| Create a 2D polygon from corridor boundary | Provides a new command for creating a closed polygon around the extents of corridor. This aids in surface creation for the corridor, as well as representing excavation limits. For more information, see To create a polyline boundary from a corridor (page 1357). | Ribbon: Home tab ➤ Create Design panel ▾ Create Boundary From Corridor |

# New Interoperability Features

| Feature | Description | Location |
|---|---|---|
| Revit building model import | Provides the ability to import an Autodesk Revit building shell and position it in the civil engineering site model using the specified units and coordinate zone. For more information, see Importing Architectural Data (page 1685). | Ribbon: Insert tab ➤ Import panel ▼ Import Building Site |
| Autodesk Revit building model display in Prospector | Displays the properties of a building site object in the Prospector tree in a Building Sites collection. The Building Site item expands to display its collections of Doors, Floors, Roofs, and Utilities. The component collections display an item list view (which contains a property column for each item in the list). <br><br> Building Sites <br> Building Site - (1) <br> Utility Connections <br> Roofs <br> Doors <br> Floors | Toolspace Prospector tab |
| Autodesk Revit building model Properties palette support | Provides access to the building object Name, Description, Style, Insertion Point X,Y,Z, and Rotation Angle properties in the AutoCAD Properties palette. | Ribbon: View tab ➤ Palettes panel ➤ Properties |
| Autodesk Revit building model grip point support | Provides grip points for the building model base point and utility connection points. | |
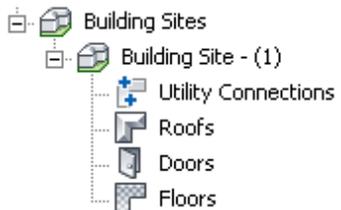
# New Intersection Features

| Feature | Description | Location |
|---|---|---|
| Create Intersection wizard | The new Create Intersection wizard automates roadway intersection design, including curb returns and widening, from specified main and side road alignments. For more information, see Creating Intersections (page 1428). | Ribbon: Home tab ➤ Create Design panel ➤ Intersection . When prompted to select an intersection, use an Osnap to select the intersection of two crossing alignments in the drawing. |

| Feature | Description | Location |
|---|---|---|
| Intersection object | A new object type, the intersection object, is created by using the wizard. It dynamically updates if its constituent parts are updated. For more information, see The Intersection Object (page 1424). | Ribbon: Home tab ➤ Create Design panel ➤ Intersection |
| Intersection grading | Provides support for the following two types of intersection grading:<br>■ Maintain the crown of the highest priority road and warp the side road cross section.<br>■ Maintain both centerlines and warp all pavements around the intersection area.<br><br>For more information, see Intersection Corridor Type (page 1434). | Ribbon: Home tab ➤ Create Design panel ➤ Intersection |
| Intersection corridors | Provides the ability to create automated intersection corridors (with all targets) and intersection-based corridor regions, using a specified assembly set.<br>Provides dynamic update of corridor regions after the intersection is created. For more information, see Creating Corridors in the Intersection Area (page 1435). | Ribbon: Home tab ➤ Create Design panel ➤ Intersection |
| Intersection profile locking | Provides the ability to lock the elevation of a lower-priority (side road) design profile to the elevation of the high priority road (main road) at the intersection location. For more information, see Changing Road Priority (page 1433). | Ribbon: Home tab ➤ Create Design panel ➤ Intersection |

# New Labels Features

**New Labels Features**

| Feature | Description | Location |
|---|---|---|
| Contour labeling masking | Provides an option in the Properties palette to mask a contour line which can improve the visibility of a contour label. For more information, see Background Mask (page 1496). | Right-click contour label ➤ Properties. In the Properties palette, on the Design tab, under Labels select Masking ➤ Contour Line Only |
| Profile and Section Label Enhancements | Provides options for staggering profile and section labels to avoid collisions in densely populated graph views. For more information, see Staggering Profile and Section Labels (page 1527). | Ribbon: Annotate tab ➤ Labels & Tables Panel ➤ Add Labels ➤ Profile View ➤ Add Profile View Labels |

| Feature | Description | Location |
|---|---|---|
| |  | |
| | Profile labels that can be staggered include Grade Break, Major/Minor Station, and Horizontal Geometry Point.<br>Section labels that can be staggered include Major/Minor Offsets and Grade Break. | |
| Band label enhancements | Provides profile and section band labels as label objects. Band labels can now be selected, dragged, and grip edited like other label objects. Band labels can also now be staggered to avoid label collisions. For more information, see Staggering Data Band Labels (page 1528). | Profile View Properties ➤ Bands tab |
| Label leader enhancements | Provides enhancements to label leaders including the ability to add and move leader vertices, to hide leader tails, and to reset the leader grip. For more information, see Managing Leader Vertices (page 1520). | Available through label leader grips |
| Major and minor increments | Provides major and minor increment options for profile and section band sets and for profile and section views.<br>This feature provides an option to match major/minor increments to vertical grid intervals. For more information, see Bands Tab (Profile View Properties Dialog Box) (page 2181). | Profile View Properties ➤ Bands tab |

# New LandXML Features

**New LandXML Features**

| Feature | Description | Location |
|---|---|---|
| LandXML version support | Provides the ability to specify which version of LandXML (1.0, 1.1, or 1.2) is used when exporting LandXML data. For more information, see Export To LandXML Dialog Box (page 1983). | Export To LandXML dialog box |
| Pipe network import settings | Provides settings to support the import of pipe network data from the Storm Sewers application. For more information, see Import Tab (LandXML Settings Dialog Box) (page 1978). | LandXML Settings dialog box, Import tab |

# New Object Projection Features

| Feature | Description | Location |
|---|---|---|
| Draw projected objects in profile and section views | Provides the ability to project an object that is visible in plan view into a profile or a section view to show the object location.<br><br><br><br>Each projected object retains a dynamic link to the source object and reflects any edits to the source. For more information, see Working with Projected Objects (page 1118). | Ribbon: Home tab ➤ Profile & Section Views panel ➤ Profile View ➤ Project Object To Profile View<br><br><br>Ribbon: Home tab ➤ Profiles & Section View panel ➤ Section View ➤ Project Object To Section View<br> |
| Grip edit projected objects | Provides the ability to grip edit projected objects in profile and section views. | |
| Label projected objects | Provides the ability to label projected objects in profile and section views. Projection label styles are managed in the Profile View and Section View label style collections. For more information, see To add a projection label (page 1038). | Ribbon: Annotate tab ➤ Add Labels ➤ Add Projected Object Label |
| Object properties exposed in Properties palette | Provides properties for projected objects in the AutoCAD Properties palette. | Ribbon: View tab ➤ Palettes panel ➤ Properties<br> |
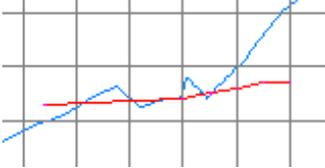
# New Parcels Features

| Feature | Description | Location |
|---|---|---|
| Parcel Slide Line and Swing Line layout and editing commands | Provides the ability to control parcel layout by specifying a minimum frontage as an offset, and/or by a minimum width and depth.<br>Certain parameters (such as area and frontage) are now required during the parcel sizing commands, so the Parcel Layout Tools dialog box expands when these commands are selected.<br>Provides graphics in the layout dialog box to illustrate the setting options. For more information, see Parcel Layout Tools (page 2009).<br><br> | Ribbon: Home tab ➤ Create Design panel ➤ Parcel ➤ Create Parcel By Layout<br> |

| Feature | Description | Location |
|---|---|---|
| Preview graphics in the drawing | Provides temporary preview graphics in the drawing to show the impact of the options and values you specify when doing parcel layout. For more information, see Creating Parcels by Subdividing (page 816). | Toolspace Settings tab ➤ Parcel ➤ Commands ➤ CreateParcelByLayout |



| Feature | Description | Location |
|---|---|---|
| Slide Angle & Direction command consolidation | Consolidates the Slide Angle and Slide Direction creation commands into a single Slide Line - Create command. For more information, see To create parcels by subdividing with the Slide Line - Create command (page 817).<br>Consolidates the Slide Angle and Slide Direction editing commands into a single Slide Line - Edit command. To edit parcels by subdividing with Slide Line - Edit command (page 821). | Ribbon: Home tab ➤ Create Design panel ➤ Parcel ➤ Create Parcel By Layout |

# New Pipe Network Features

**New Pipe Network Features**

| Feature | Description | Location |
|---|---|---|
| Multiple pipe network support | Provides the ability to import multiple networks from Storm Sewers. For more information, see Importing Pipe Network Data from an .stm File (page 1278). | Ribbon: Insert tab ➤ Import panel ➤ Storm Sewers |
| Part matching logic on import | Provides new part matching logic when importing Storm Sewers data into AutoCAD Civil. During import, the part names are used as the unique identifier for pipe networks.<br>If a line or inlet/junction does not have a name during import, it is named based on the naming template within the pipe network feature settings.<br>During import, if there is a part size that does not exactly match the AutoCAD Civil parts list, the closest matching larger size is used. For more information, see Preparing for Migrating Storm Sewers Data (page 1275). | Toolspace Settings tab ➤ Pipe Network ➤ Commands ➤ ImportStormSewerData, Storm Sewers Migration Defaults settings |
| Export pipe networks to Storm Sewers | Provides the ability to export Pipe Networks to Storm Sewers so pipe networks can be analyzed. For more information, see Exporting Pipe Network Data to an .stm File (page 1279). | Ribbon: Output tab ➤ Export panel ➤ Export To Storm Sewers |

| Feature | Description | Location |
|---------|-------------|----------|
| Parts list with all catalog parts | Provides a command to create a parts list that contains all the parts that exist in AutoCAD Civil.<br>This new command can be useful if you are migrating pipe network data between AutoCAD Civil and the Storm Sewers application, for example. For more information, see Creating a Full Parts List (page 1239). | Ribbon: Home tab ➤ Create Design panel ➤ Create Full Parts List |
| LandXML Import settings | Provides Pipe Network Import Settings in the existing LandXML Settings dialog box. This option is to support the import of pipe network data from the Storm Sewers application. For more information, see Import Tab (LandXML Settings Dialog Box) (page 1978). | Toolspace Settings tab ➤ right-click drawing collection item ➤ Edit LandXML Settings |
| Hydraulic Grade Lines and Energy Grade Lines | Provides the ability to display and label hydraulic grade lines (HGL) and energy grade lines (EGL) for pipes and structures in profile views.<br>The Pipe Properties dialog box and the Structure Properties dialog box now have items for HGL and EGL on the Part Properties tabs. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280). | Pipe Properties dialog box ➤ Part Properties tab, Geometry values<br>Structure Properties dialog box ➤ Part Properties tab, Geometry values |
| Display of HGL/EGL components in profile view | Provides display options for HGL and EGL components in the pipe and structure style dialog boxes. To represent the hydraulic grade line and energy grade line in a profile view, you must enable their display in the pipe and structure styles.<br>The visibility for these components is off by default. For more information, see Viewing Hydraulic Data in Profile Views (page 1282). | Pipe Style dialog box ➤ Display tab ➤ Profile View Direction, Hydraulic Grade Line and Energy Grade Line components<br>Structure Style dialog box ➤ Display tab ➤ Profile View Direction, Hydraulic Grade Line and Energy Grade Line components |
| HGL/EGL Labels | Provides the ability to label hydraulic grade lines and energy grade lines for pipes and structures in profile view. For more information, see Labeling Hydraulic Properties in Pipe Networks (page 1270).<br>The following new label properties are available for pipes:<br>■ Hydraulic Grade Line Up<br>■ Hydraulic Grade Line Down<br>■ Energy Grade Line Up<br>■ Energy Grade Line Down<br><br>The following new label properties are available for structures:<br>■ Hydraulic Grade Line<br>■ Energy Grade Line | Pipe and Structure label styles |

| Feature | Description | Location |
|---|---|---|
| New Pipe Network Parts Catalog Content | Enhancements have been added to the pipe network parts catalog. For example, the following new part types are now available: | Toolspace Settings tab ➤ Pipe Network ➤ Parts List ➤ Add Part Family |

**Circular pipes**

- Corrugated Metal Pipe
- HDPE Pipe
- Corrugated HDPE Pipe

**Elliptical pipes**

- Concrete Horizontal Elliptical Arch Pipe

**Structures (Inlet\Outlet)**

- Concrete Flared End Section
- Concrete Rectangular Winged Headwall
- CMP Rectangular End Section

In addition to these new part types, various new sizes have been added to existing part families.

# New Profile Features

**New Profile Features**

| Feature | Description | Location |
|---|---|---|
| Profile hatching | Provides the ability to add hatch patterns to a profile view that contains multiple profiles. The hatch patterns can be used to show cut, fill, multiple boundaries, and quantity takeoff in the profile.<br>You can specify a Shape Style for each hatch area, which defines which hatch pattern is used. For more information, see Adding Hatch Areas to a Profile View (page 1116). | Create Profile View wizard, Profile View Properties dialog box |

# New Surfaces Features

**New Surfaces Features**

| Feature | Description | Location |
|---|---|---|
| Clip boundaries | Enables data clip boundaries to be applied to imported surfaces including TIN, LandXML, grid, or corridor surfaces. For more information, see Applying a Data Clip Boundary to Imported Surfaces (page 614). | Ribbon: Surface context tab ➤ Modify panel ➤ Add Data ➤ Boundaries |
| AutoCAD polylines from contour data | Provides a command to insert contours that are used in a surface into a drawing as polylines.<br>If a polyline has been added as contour data, you can run the Insert To Drawing command and one or more AutoCAD polylines are added to the drawing. In addition, the record of the | ■ On the Toolspace Prospector tab, select the Contour collection in a surface definition to display the list view at |

| Feature | Description | Location |
|---|---|---|
| | contours (in the surface operations list) is converted from a list of coordinates to a reference to the new polylines.<br>For more information, see Inserting Contour Data Into a Drawing (page 634). | the bottom of Prospector.<br>■ Right-click a contour in the list and click Insert To Drawing. |
| New command and feature settings | ■ SurfaceExportToDEM command setting: Null-elevation and grid spacing.<br>■ AddSurfaceBreaklines/AddSurfaceContours command settings: Weeding and supplementing factors and mid-ordinate distance.<br>■ Rebuild Surface feature setting: Value in Surface Defaults collection for automatic rebuild of a surface.<br><br>For more information, see Edit Feature Settings - Surface Dialog Box (page 2329). | Toolspace Settings tab ➤ Surface collection ➤ Commands collection ➤ right-click command name ➤ Edit Command Settings |

# New Survey Features

**New Survey Features**

| Feature | Description | Location |
|---|---|---|
| Import survey data wizard | Provides a wizard to streamline the import of a field book file, LandXML file, point file, or points from the drawing, into the survey database. For more information, see Import Survey Data Wizard (page 239). | Ribbon: Home tab ➤ Create Ground Data panel ➤ Import Survey Data |
| Linework code sets | Provides the ability to define a linework code set that can be used for importing a field book file, LandXML file, point file, or points from the drawing.<br>You can create a named field coding convention, where <begin>, <continue>, <end>, <line/curve commands>, and so on, are defined. You can define one-to-many conventions, where a specific convention can be specified at import time.<br>Linework code sets are grouped in a Linework Code Sets collection on the Toolspace Survey tab.<br>For more information, see Linework Code Sets (page 230). | Toolspace Survey tab ➤ right-click Linework Code Sets ➤ New |
| Import Field Book file | Provides the ability to import a field book and specify which named import convention and feature code table to use.<br>The survey points and line features are added to the drawing and to the survey database. For more information, see Importing a Field Book File (page 241). | Toolspace Survey tab ➤ Survey Databases ➤ <named database> ➤ Networks ➤ right-click <named network> ➤ Import ➤ Import Field Book |
| Import ASCII point file | Provides the ability to import an ASCII point file containing a Point ID, X-coordinate, Y-coordinate, Z-coordinate, and Description.<br>To import an ASCII point file, you specify the point file format, the import convention, and the feature code table. The survey points and line features are added to the drawing and to the | Ribbon: Home tab ➤ Create Ground Data panel ➤ Import Survey Data |

| Feature | Description | Location |
|---|---|---|
| | survey database. For more information, see To import a point file (page 248). | |
| Import points from drawing | Provides the ability to import points from the current drawing into the survey database and network as either control points or non-control points, and provides an option to perform line-work processing. For more information, see Importing Points From a Drawing (page 249). | Toolspace Survey tab ➤ Survey Databases ➤ <named database> ➤ Networks ➤ right-click <named network> ➤ Import ➤ Import Points From Drawing |
| Import LandXML | Provides the ability to import a LandXML file using a named import convention. For more information, see Importing Survey LandXML Data (page 244). | Toolspace Survey tab ➤ Survey Databases ➤ <named database> ➤ Networks ➤ right-click <named network> ➤ Import ➤ Import Field Book |
| Import Events item in database collection | Provides the ability to update data after importing from a field book or ASCII File. Upon completion of the import process, you can edit the appropriate point descriptions and reprocess the data using the current import convention and feature code table, replacing erroneous line features with the corrected line features. You can perform these steps iteratively until satisfied that field coding errors have been minimized. For more information, see Import Events (page 249). | Toolspace Survey tab ➤ Survey Databases ➤ <named database> ➤ Import Events |

# Changes to Drawing Templates

**Included Templates**

| Feature | Description |
|---|---|
| Templates included in the product | The "Extended" templates included in previous releases are now named as follows:<br>■ _AutoCAD Civil 3D (Imperial) NCS<br>■ _AutoCAD Civil 3D (Metric) NCS<br><br>Some styles in the "LDT" template that was included in previous releases have been moved into these templates. The "Base" and "LDT" templates are no longer included in the product. |

**Drawing Settings**

| Feature | Description |
|---|---|
| Drawing Settings ➤ Object Layers tab | **New layer settings**<br>■ Building Site<br>■ Intersection<br>■ Intersection-Labeling<br>■ Grading-Labeling<br>■ Parcel-Labeling |

| Feature | Description |
|---|---|
| | ■ Parcel Segment-Labeling |
| | **Changed layer settings** |
| | ■ Pipe (now C-STRM-PIPE; was C-STRM) |
| | ■ Pipe Network Section (now C-STRM-SCTN; was C-STRM) |
| Drawing Settings ➤ Ambient Settings tab ➤ General | Driving Direction |

**General Settings**

| Feature | Description |
|---|---|
| Multipurpose Styles ➤ Shape Styles | Multiple Boundary Material style |

**Projection Object Settings**

| Feature | Description |
|---|---|
| Multipurpose Styles | Projection Styles collection |
| Feature line style | ■ Profile tab<br>■ Section tab<br>■ Profile and Section View Directions on Display tab |
| Survey figure style | ■ Profile tab<br>■ Section tab<br>■ Profile and Section View Directions on Display tab |
| Multipurpose Styles ➤ Marker Styles | ■ Projection in Section Basic<br>■ Projection in Section Buildings<br>■ Projection in Section Curb<br>■ Projection in Section Fences<br>■ Projection in Section Road Centerline<br>■ Projection in Section Treeline |
| Point style | Profile and Section View Directions on Display tab |
| Label styles | ■ Profile View Projection Label<br>■ Section View Projection Label |
| Profile View feature settings and command settings for ProjectObjectsToProf | ■ Projection Style<br>■ Profile Projection Label Style<br>■ Default Projection Label Placement |

| Feature | Description |
| --- | --- |
| Section View feature settings and command settings for ProjectObjectsToSect | ■ Projection Style<br>■ Section Projection Label Style<br>■ Default Projection Label Placement |

**Parcel Settings**

| Feature | Description |
| --- | --- |
| CreateParcelByLayout command settings | **Parcel Sizing Properties**<br>■ Parcel Selection Method<br>■ Minimum Area<br>■ Use Minimum Frontage At Offset<br>■ Frontage Offset<br>■ Minimum Width<br>■ Minimum Depth<br>■ Use Maximum Depth<br>■ Maximum Depth<br>■ Multiple Solution Preference |
| CreateParcelByLayout command settings | ■ Preview Graphics |

**Grading Settings**

| Feature | Description |
| --- | --- |
| CreateFeatureLineFromAlign command settings | Spiral Tessellation Factor |

**Alignment Settings**

| Feature | Description |
| --- | --- |
| Alignment feature settings | ■ Default Name Format ➤ Offset Alignment Name Format<br>■ Dynamic Alignment Highlight Options |
| CreateAlignmentEntities command settings | Alignment Type Option ➤ Alignment Type |
| AddWidening command settings | Widening Options |
| CreateOffsetAlignment command settings | Dynamic Offset Alignment Options |
| Alignment styles | ■ Intersection Basic style<br>■ Offsets style |

**Profile and Profile View Settings**

| Feature | Description |
| --- | --- |
| Station and grade break labels | Staggering options |

| Feature | Description |
|---|---|
| Profile View feature settings ➤ Default Styles | ■ Cut Area Shape Style<br>■ Fill Area Shape Style<br>■ Multiple Boundary Area Shape Style |
| Profile View feature settings ➤ Default Name Format | ■ Cut Area Name Template<br>■ Fill Area Name Template<br>■ Multiple Boundary Area Name Template |
| Profile View styles ➤ Display tab | Profile Hatch setting |
| Band Set styles | ■ Major Interval<br>■ Minor Interval<br>■ Stagger Labels<br>■ Stagger Line Height<br>■ Check box for match major/minor increments to vertical grid intervals |
| Band label styles | Addition of Dragged State and General tabs for all label styles in all bands |

**Section View Settings**

| Feature | Description |
|---|---|
| Band Set styles | ■ Major Interval<br>■ Minor Interval<br>■ Stagger Labels<br>■ Stagger Line Height<br>■ Check box for match major/minor increments to vertical grid intervals |
| Band label styles | Addition of Dragged State and General tabs for all label styles in all bands |

**Pipe Network Settings**

| Feature | Description |
|---|---|
| Feature settings | Default Styles ➤ Structure Default Style (now Storm Sewer Manhole; was Sanitary Sewer Manhole) |
| Feature settings and the following command settings:<br>■ EditInStormSewers<br>■ CreateNetworkPartsListFull<br>■ ExportStormSewerData<br>■ ImportStormSewerData | Storm Sewers Migration Defaults |

| Feature | Description |
| --- | --- |
| Command settings for the following commands<br>■ EditInStormSewers<br>■ ImportStormSewersData<br>■ ExportStormSewerData | ■ Storm Sewers migration default settings |
| Partial Catalog Parts List | Contains a revised version of the Storm Sewers parts list |
| Pipe style ➤ Display tab (for Double Line (Sanitary) and Double Line (Storm) styles) | ■ Hydraulic Grade Line<br>■ Energy Grade Line |

### Corridor Settings

| Feature | Description |
| --- | --- |
| Feature settings | Region Highlight Graphics category |

### Intersection Settings

| Feature | Description |
| --- | --- |
| Feature settings | All new |
| Command settings | All new |
| Intersection style | Intersection Marker Style |
| Label style | Intersection Label |

### Building Site Settings

| Feature | Description |
| --- | --- |
| Building Site style | All new |
| Command settings | All new |

### Plan Production Template Settings

| Feature | Description |
| --- | --- |
| Plan Production templates | No changes |

# Feature Overview

<div style="text-align: right; font-size: 3em; font-weight: bold;">2</div>

This section explains and illustrates the concepts underlying AutoCAD Civil features.

Each feature section has one or more corresponding illustrations to reinforce the concepts introduced in that section.

## Points

Points are the basic building blocks used to identify entities in land development plans such as ground locations and design elements. Each point is uniquely identified and has properties that include information such as northing, easting, elevation, and description.

You can insert or import points from different sources and in different formats. For example, you can define sets of description keys to help organize the insertion of points. Description keys use the raw description of incoming points to control the creation of each drawing point, including the following actions:

■ Assigning the point to a specific layer

■ Assigning a point style

■ Assigning a point label style

■ Translating the raw description into a full description

■ Rotating or scaling the point symbol

Points can also be created directly in the drawing, using a variety of methods.

You can use point groups to organize points and to control their appearance in a drawing. Point groups reference the point data directly and are responsible for drawing the points. Point groups have an override that forces all points in the group to use both that group's preferred point style and point-label style, rather than the styles assigned to the individual points.

Storm Manholes Point Group

Road Centerline point group

Pond Point Group

Landscape Trees point group
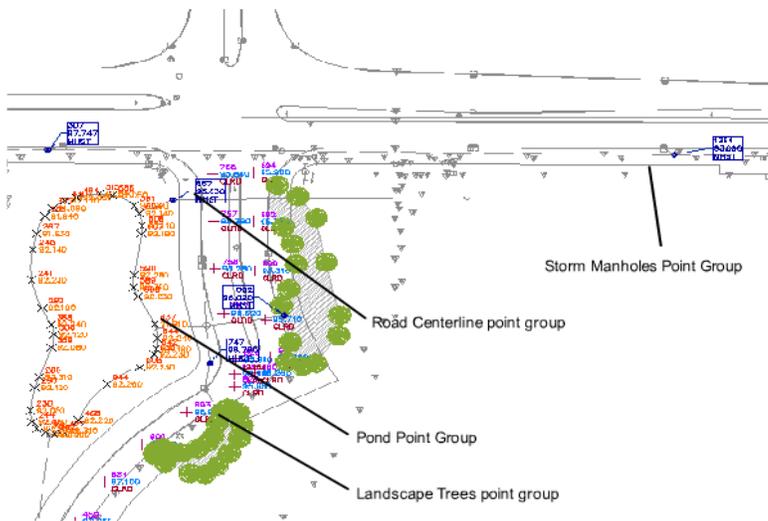
**Point groups showing different uses of point styles**



Point label style
Manholes includes an
arrow and leader line.

Point style Tree-20ft Pine has labels turned off.

Pond point group uses the Number and Elevation
Only point label style.
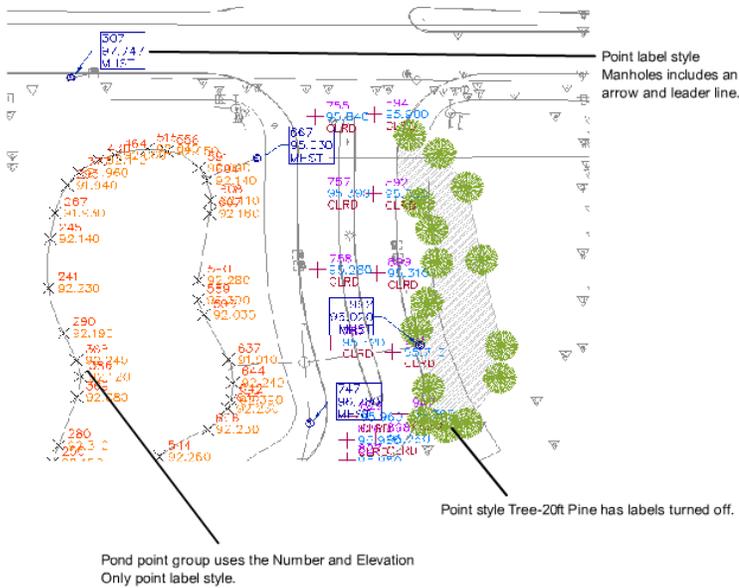
**Examples of point label styles**

| For information about…. | See... |
|---|---|
| Working with Points | Understanding Points (page 407) |
| Workflow | Points Workflow |
| Tutorials | Points Tutorials |

# Surfaces

You can work with two types of surfaces: TIN (triangulated irregular network) and grid. For each of these types, you can create volume surfaces, which are differential surfaces created from two existing surfaces. Surface styles define the appearance of any surface. They can also be used to control the visibility of any analysis that has been performed on that surface. Watersheds can be drawn on the surface, with information about the type of drainage area and where each area drains to.

Boundaries define the visible area of a surface. Only the area within the boundary is included in calculations, such as for total area and volume. You can also define masks to hide or show parts of a surface for editing or presentation purposes, while still including that area in calculations.
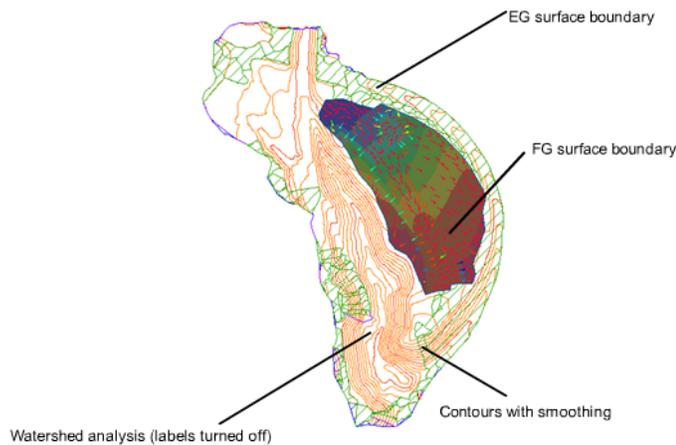


**Surfaces displaying boundaries, contours, and elevation analysis**

Breaklines are used on TIN surfaces to define linear features that triangles cannot cross, such as retaining walls or streams. Breaklines affect triangulation of the surface.

You can define different sets of contours, for example, for different intervals. Smoothing is provided for the surface object as a whole, which gives better results than simply smoothing the contours. In AutoCAD Civil, the build process for surfaces is incremental. Whenever data is added or corrected, the surface is updated. Each surface has a definition list. This list contains all the operations performed on the surface. By turning the operations on and off, you can return a surface to a previous state or modify it to support different types of analysis.
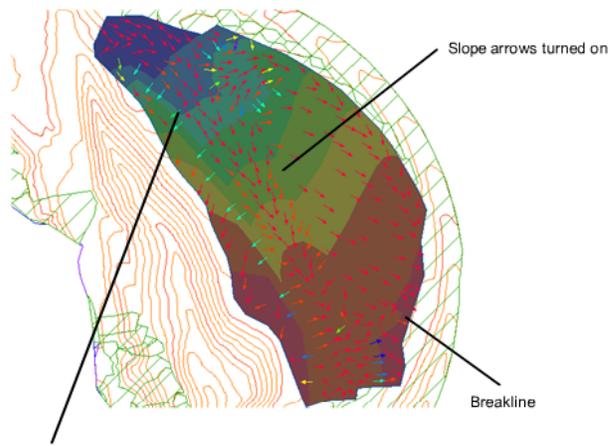
Surfaces displaying slope arrows elevation analysis

| For information about…. | See... |
| --- | --- |
| Working with Surfaces | Surfaces (page 599) and Surfaces Dialog Boxes (page 2329) |
| Workflow |  Surfaces Workflow |
| Tutorials |  Surfaces Tutorials |
| Best Practices |  Surface Data |

# Grading

You can use the grading tools to design finished graded surfaces. Gradings are objects that have their own properties and behavior like other AutoCAD Civil objects. You create a grading by selecting a base line in the drawing, and then specifying a projection method and target, for example, grade at 3:1 to meet an existing surface. Save time and effort by predefining such values and saving them as grading criteria. Subsequent gradings you create will use the current criteria. You can also create named grading styles that combine specific display properties, such as colors. In addition to using the current criteria, any grading you create uses the current style.

A grading normally consists of a face bounded by a base line, a target line, and several projection lines. The base line can be any open or closed figure from which you want to project the grading. It can be a feature line or a lot line. A feature line is any linear feature in the drawing, such as a ridge line, building footprint, or the bottom of a swale. The target for the grading can be a surface, a distance, or an elevation (absolute or relative).
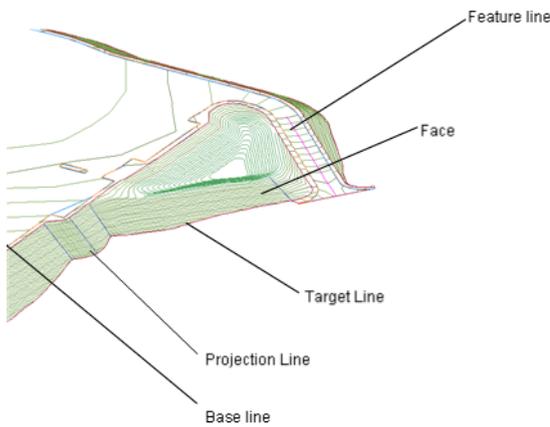


**Line components of a grading object**

Each site can include grading groups, which bundle individual gradings into named sets. Before creating a grading, you must create a new grading group, or select one that already exists. A surface can be created from a grading group, and a grading group can be pasted into a surface. The surface will then be updated if you change the grading.

After you create a grading group, volume tools within AutoCAD Civil show you the amount of cut and fill required for the grading design. You can raise or lower the grading group incrementally to adjust volume requirements. You can also change the elevation of points along a grading base line, change the grade of a base line, or modify the grading criteria.

Gradings using different criteria

Criteria uses slope of 3:1 with a surface target
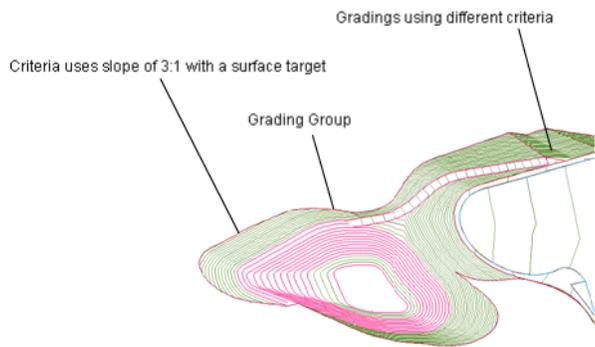
Grading Group

**Grading criteria specifies the grading method**

| For information about…. | See… |
| --- | --- |
| Working with Grading | Grading (page 729) and Grading Dialog Boxes (page 1875) |
| Workflow | Grading Workflow |
| Tutorials | Grading Tutorials |
| Best Practices | Grading Best Practices |

# Parcels

AutoCAD Civil provides a site topology that includes intelligent parcel objects. Each parcel is an independent object that usually represents real estate parcels or lots. You can also import parcels as simple polylines and then convert them to parcel objects. Parcels consist of a series of segments that can be edited individually. Editing parcel segments dynamically updates the parcel properties. Deleting a segment can result in one parcel merging with another.
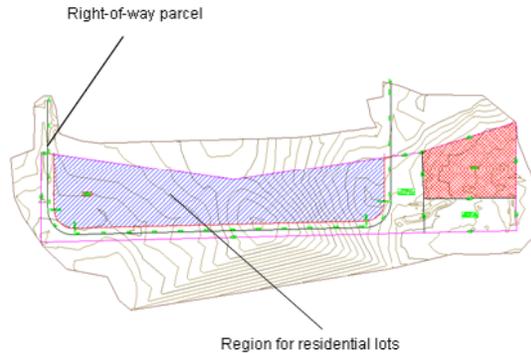


**Parcel region before subdivision into individual lots**

Each site contains one collection of parcels. The hierarchy of parent site and child parcels is managed on the Prospector tab in Toolspace. The parent site represents the original area to be subdivided. By creating a number of separate sites, you can manage large projects with many lots grouped in separate blocks.

You can create parcels one at a time or as a group, with settings for the minimum area and minimum frontage as well as the minimum/maximum width and depth of each parcel. Parcel layout tools give you precise control over parcel area and the angle of each lot line.

Parcel styles determine the appearance of the parcel, including fill patterns for the area and linetypes for the segments. There are separate label styles for parcel areas and parcel segments.

Parcel style uses area fill

Area Selection Label

Segment Label
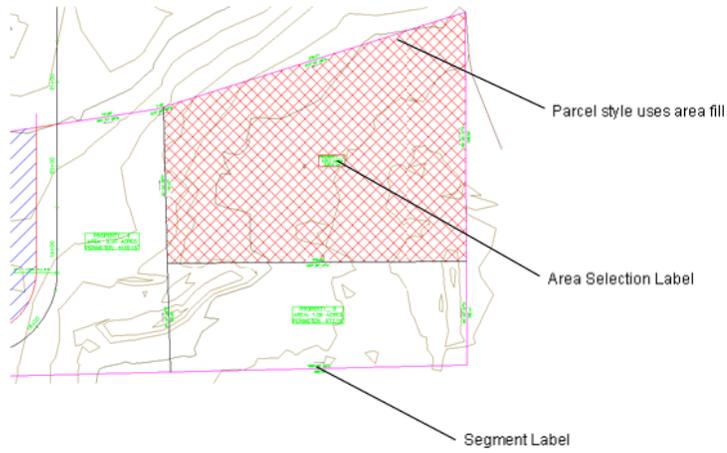
**Parcel display style and two types of label**

| For information about…. | See... |
|---|---|
| Working with Parcels | Parcels (page 807) and Parcels Dialog Boxes (page 1999) |
| Workflow |  Parcels Workflow |
| Tutorials |  Parcels Tutorials |
| Best Practices |  Parcels |

# Alignments

Horizontal alignments are used to represent roads and other linear objects. Alignments can include lines, curves, and spirals, which can be joined by constraints. When you edit an alignment, for example, by dragging a grip, the components of the alignment can maintain tangency to one another. You can create alignments from existing polylines, or by using AutoCAD Civil alignment layout tools. You can edit an alignment either by grip editing or by using the edit commands. Some of the creation and editing options are as follows:

- Draw a series of tangents, then add curves or spiral-curve or spiral-line groups at the PIs (Points of Intersection).

- Draw single lines, curves, and spirals.

- View and edit the dimensions of alignment components.

- Apply superelevation to the curves along an alignment.

Use station equations to change station-distance references at any point without physically changing the geometry of the alignment.
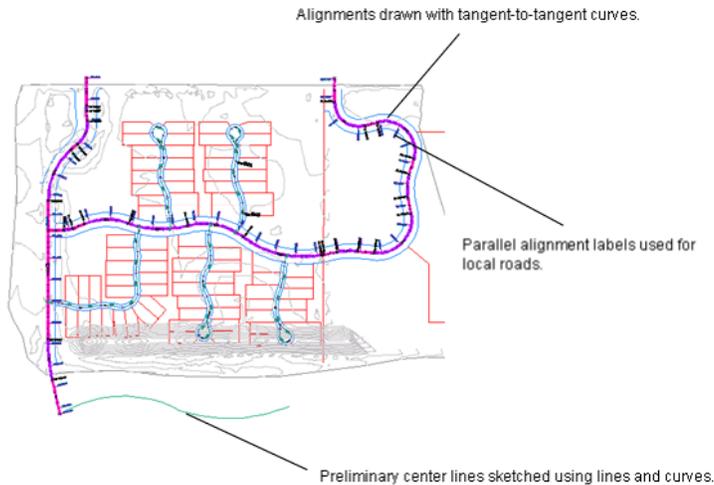


**Alignment lines, curves, and labels**

Station reference point at start of alignment.

Major road uses different line and label styles from local roads.

Station Label

Design speed label
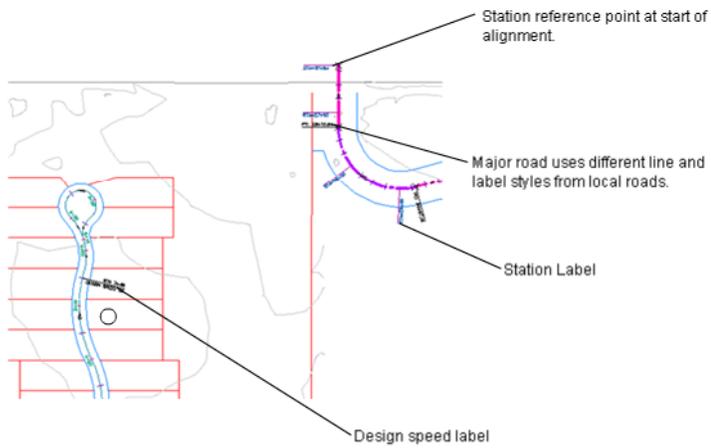
**Examples of alignment labels**

| For information about…. | See… |
| --- | --- |
| Working with Alignments | Alignments (page 855) and Alignment Dialog Boxes (page 1735) |
| Workflow | Alignments Workflow |
| Tutorials | Alignments Tutorials |
| Best Practices | Alignments |

# Profiles

Profiles (also known as vertical alignments), are derived from horizontal alignments in the drawing. There are two types of profiles. Surface profiles, often called existing ground (EG) profiles, are extracted from a surface. Layout profiles, often called finished grade (FG) profiles, represent a designed surface such as a road. A profile can be dynamic, in which case it is linked to a surface to reflect updates to the surface or the horizontal alignment. It can also be static, to preserve a record of a surface at a particular time.You create static profiles in various ways, such as importing a text file, importing an XML file, or by using the Profile Layout Tools dialog box.

Edit a profile by grip editing or by using the edit commands in the Profile Layout Tools dialog box.

Profiles are displayed in graphs called profile views. Profile views are separate objects that have their own sets of styles. You can add data bands to annotate the profile view with stations and elevations, horizontal geometry points, or other useful data. You can save several data bands in a set that can be easily applied to other profile views.
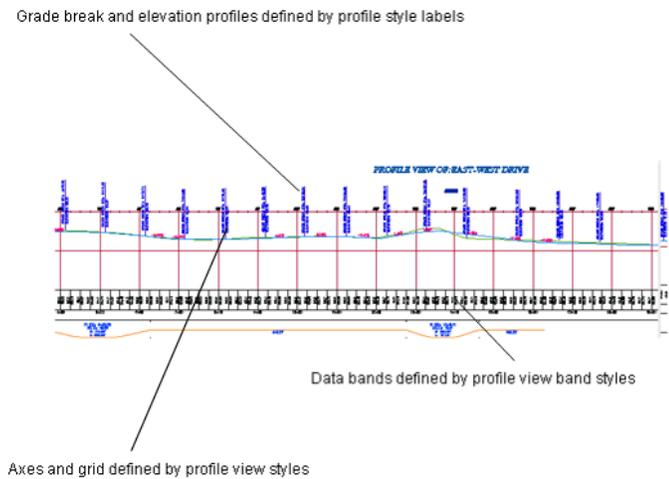


**Profile view displaying surface and layout profiles**

Profile label shows a tangent grade

Surface and layout profiles
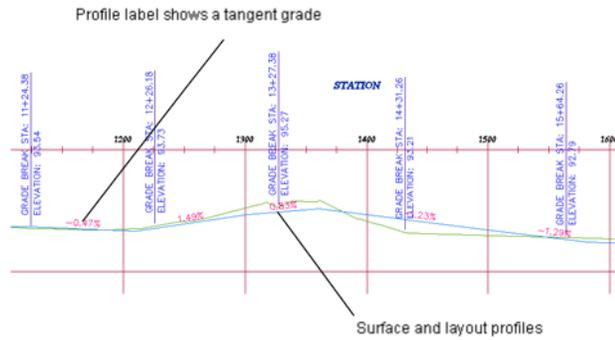
**Closer view of surface and layout profiles**

| For information about…. | See… |
| --- | --- |
| Working with Profiles | Profiles (page 1019) and Profiles Dialog Boxes (page 2139) |
| Workflow |  Profiles Workflow |
| Tutorials |  Profiles Tutorials |
| Best Practices |  Simplify Profile and Section View Styles |

# Sections

Sections, or cross sections, are cut across a linear feature to a specified distance on the left and right of a center line. Sections are typically sampled at given stations along a horizontal road alignment. Edits to an alignment update the sections as well.

Sections are located on sample lines that cut across the alignment. Sample lines have their own styles and can be labeled. A set of sample lines makes up a named collection called a sample line group.

Sample lines belong to the sample line group



Stations and sample-line names are displayed by the sample line group label style
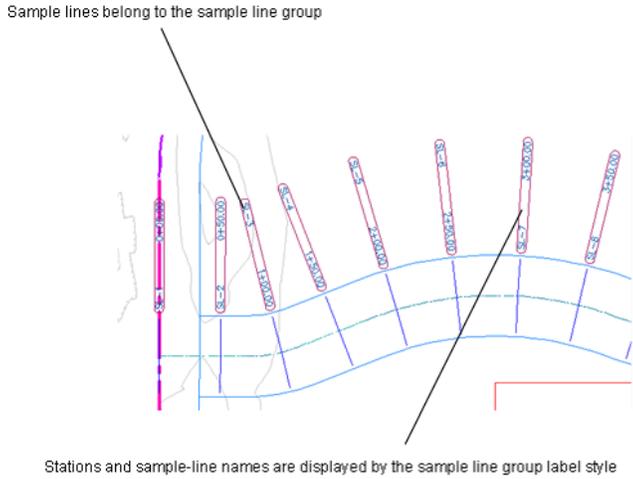
**A group of sample lines**

Sections are displayed in a graph called a section view. A section view is very similar to a profile view. It consists of a grid or graph with attributes that are controlled by section view styles. You can also display data bands above or below the section view. You can plot individual sections for a specific sample line, or all the sections for a group of sample lines. To plot multiple sections, use a paperspace viewport to set up the sheet.

Sections, like alignments and profiles, support label sets. Label sets enable you to save and apply an unlimited number of different types of labels.
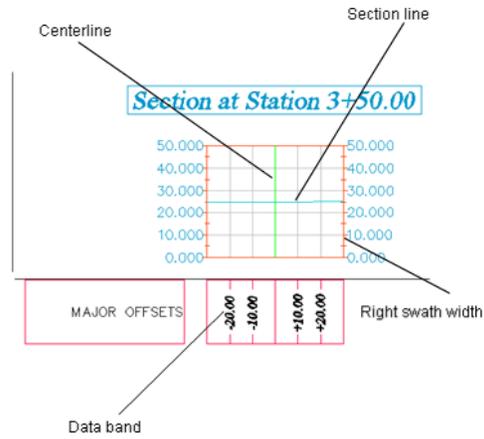
**Section view of a sample line**

| For information about…. | See... |
| --- | --- |
| Working with Sections | Sections (page 1123) and Sections Dialog Boxes (page 2229) |
| Workflow | Sections Workflow |
| Tutorials | Sections Tutorials |
| Best Practices | Simplify Profile and Section View Styles |

# Pipe Networks

You can use the pipe networks feature to design and model networks that represent the flow and function of a utility system, such as a storm or sanitary sewer.

You build a model of a pipe network using individual parts, much like a real-world pipe network, connecting pipes in your network either with or without using structures, such as manholes or catch basins, and adding inlet or outlet structures, such as headwalls, to mark the terminating end of a pipe run. After you have created your initial network design, you can view and edit network parts in virtually unlimited ways within plan view or in a profile view. You can also view the pipe network parts in a section.

Preset design rules control the slope of pipes, their depth relative to a surface, and the sizing of the structures that connect the pipes. This is useful when you are designing a gravity system, such as a storm or sanitary sewer.

### Pipe Network Viewports

This illustration displays three viewports: the pipe network in plan view, the profile view of some of the network pipes and structures, and the same parts in a section view.

The left pane shows a pipe network segment made up of two manhole structures connected by pipes. The network parts have been placed at a specified offset from an alignment, and are drawn in a profile view created using the same alignment (upper-right pane). A sample line placed at station 7+71 was used to create a section view (lower-right pane) that shows a cross-section view of the pipe network parts located at the station.

You can edit the positions of the pipe network parts manually by using editing grips in either plan, profile, or section view, or by directly editing their properties.
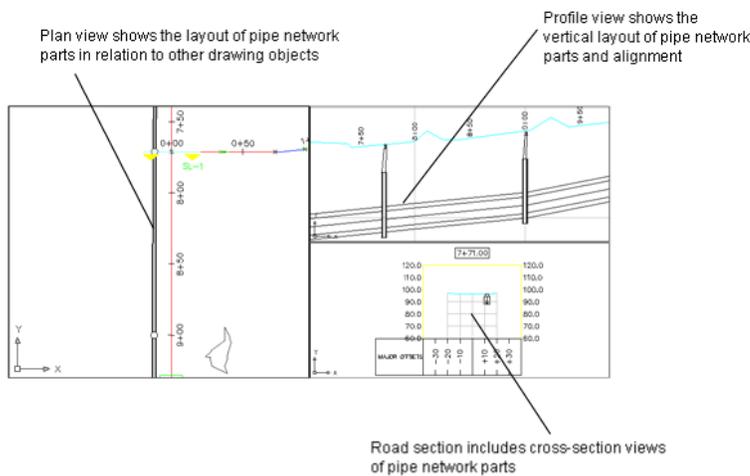


**Layout, profile, and section views of a pipe network segment**
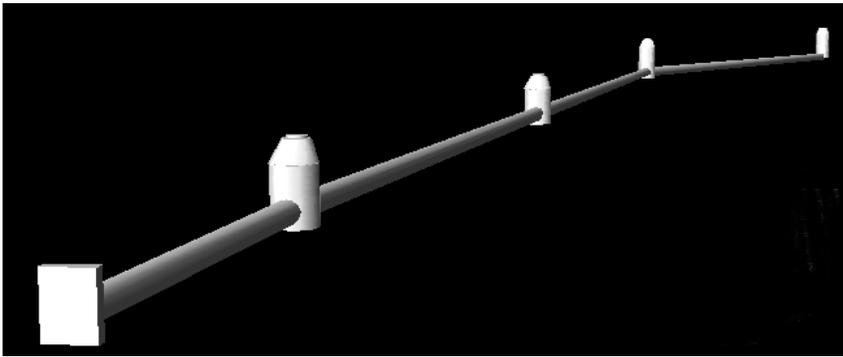
**3D view of pipe network segment**

**Pipe Parts**

A pipe network is constructed from various pipes and structures that appear in a parts list. You can see the available lists on the Toolspace Settings tab by expanding the Pipe Network ➤ Parts Lists collection.

AutoCAD Civil includes a utility called Part Builder, which you can use to edit the dimensions of parts and create new ones. You can create a new part by saving an existing one with a new name, and then modifying its dimensions in various tables.
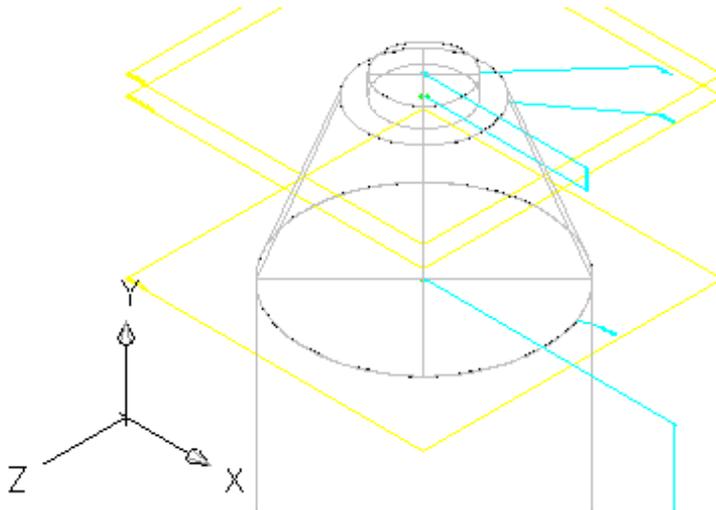


**Detail from a Part Builder edit window for a concentric cylindrical structure.**

| For information about…. | See... |
|---|---|
| Working with Pipe Networks | Pipe Networks (page 1185) and Pipe Networks Dialog Boxes (page 2023) |
| Workflow | Pipe Networks Workflow |
| Tutorials | Pipe Network Tutorials and Part Builder Tutorials |
| Best Practices | Pipe Networks |

| For information about…. | See... |
|---|---|
| Working with Part Builder | Understanding Part Builder (page 1287) and Creating Parts with Part Builder (page 1307) |
| Working with Hydraulic and Hydrology Data | Hydraulics and Hydrology Feature Overview (page 1190) |
| Best Practices | Hydraflow Extensions |

# Corridors

Corridors are used for roads and similar structures that follow a designed route across the terrain.

A corridor is a detailed three-dimensional design that combines data from a horizontal alignment, a profile view, and an assembly. The corridor can automatically reflect changes to any of the parent data.

Subassembly objects such as road lanes, curbs, and shoulders form the building blocks for an assembly. When you apply the assembly to an alignment and a profile, a corridor is generated in three dimensions. At each point along its linear path, the corridor adapts to conditions, such as superelevation and cut or fill requirements.

### Intersections

You can use intersection design features to automatically create flexible 3D models of corridor intersections. Intersection objects are created from various object data, including alignments, profiles, corridors, assemblies, subassemblies, and surfaces.

### Corridor Viewports

This illustration displays three viewports. The viewports help you see the object relationships.

- In the left pane, the horizontal alignment starts at the top of the corridor and runs to the bottom.

- In the profile view, station 0 is on the left side, and green vertical lines across the grid show the start and end points of the horizontal curves.

- In the assembly view, the corridor assembly includes two road lanes, a curb on the left, a sidewalk on the right, and side slopes on each side. The side slope subassembly (BasicSideSlopeCutDitch) is designed to create a ditch only on cut slopes, and the ditches are visible along the corridor.

You can use the Object Viewer to see the corridor in a three-dimensional view that shows the cut and fill regions. Click the corridor, then right-click and click Object Viewer. Click near the bottom of the Object Viewer window, then click and move the cursor up to tilt the corridor. If you want to pan or zoom the display, right-click inside the Object Viewer window.

Corridor shows the design in relation to the surface

Profile shows the vertical layout and the surface

Assembly is a template for corridor cross-sections
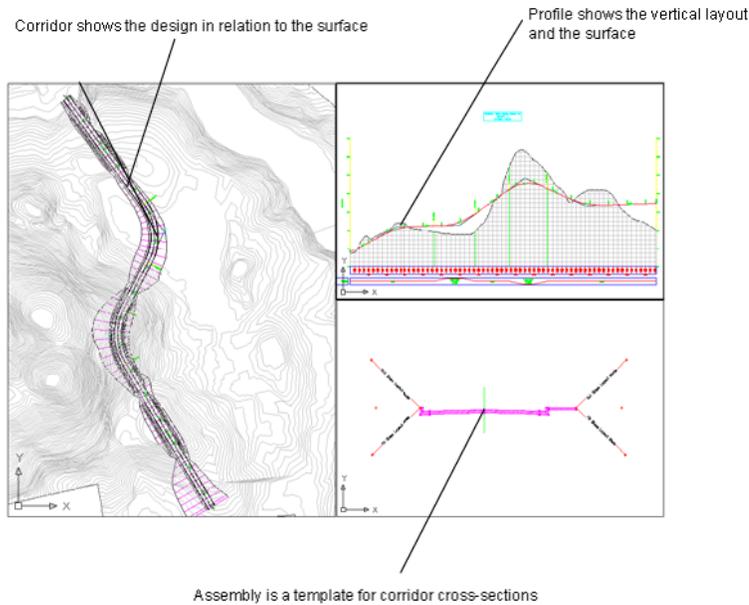
**Elements of a corridor design**

## Assemblies and Subassemblies

A corridor is created from one or more assemblies, which are standard roadway cross-sections. You design an assembly from subassemblies, such as lanes, curbs, shoulders, and ditches. The subassemblies are provided in a set of catalogs.
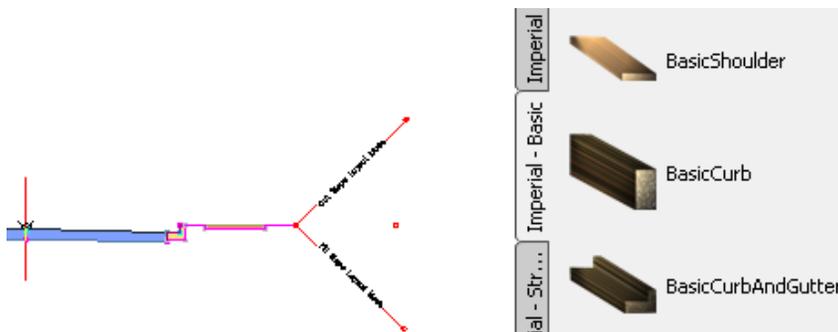


**Roadway assembly (left), with subassemblies shown in a tool palette (right)**

Each subassembly has a defined cross-section, and some subassemblies automatically adapt to their location. For example, the slope of a road lane changes as superelevation is applied, and a side slope automatically creates either a cut or fill slope, depending on the relative elevation of the existing surface.

The dimensions of a subassembly, such as the width of a lane or the height of a curb are stored as properties.

You can also create custom subassemblies from AutoCAD polylines. In this case, you must also specify the subassembly behavior within an assembly and in the process of corridor creation. You can define custom subassemblies and their behavior using .NET scripts, then use the AutoCAD Civil COM application programming interface (API) to link these to the main application.

A subassembly definition references point, link, and shape codes. Points are the vertices of the subassembly, and they can be attachment points for adjacent assemblies. Links are the line segments or curves between the points. Shapes are two-dimensional polygons that represent the cross-sectional shape of the subassembly. The following figure shows a coding diagram for a Basic Curb and Gutter subassembly:
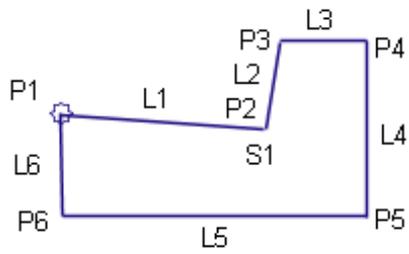
| For information about…. | See... |
| --- | --- |
| Working with Corridors | Understanding Corridor Modeling (page 1327) |
| Workflow |  Corridor Modeling Workflow |
| Tutorials |  Corridors Tutorials |
| Best Practices |  Corridor Design |
| Working with Assemblies and Subassemblies | Assemblies and Subassemblies (page 1377) |
| Tutorials |  Corridor Assembly Tutorials |
| Working with Intersections | Intersections (page 1423) |
| Tutorials |  Intersection Tutorials |

# Survey

You can use the AutoCAD Civil survey tools to import, analyze, adjust, and convert survey data. The Survey tools can also automate symbol placement and linework creation to facilitate the conversion of survey data into AutoCAD Civil objects.

Survey data is organized into databases, whose data is visible in Toolspace, on the Survey tab. The survey databases can either be integrated with your Autodesk Vault project management database, or stored locally.

During the field survey data collection process, you can use a coding convention established in AutoCAD Civil to automate the conversion of field survey data, collected with data collection equipment, into AutoCAD Civil linework.
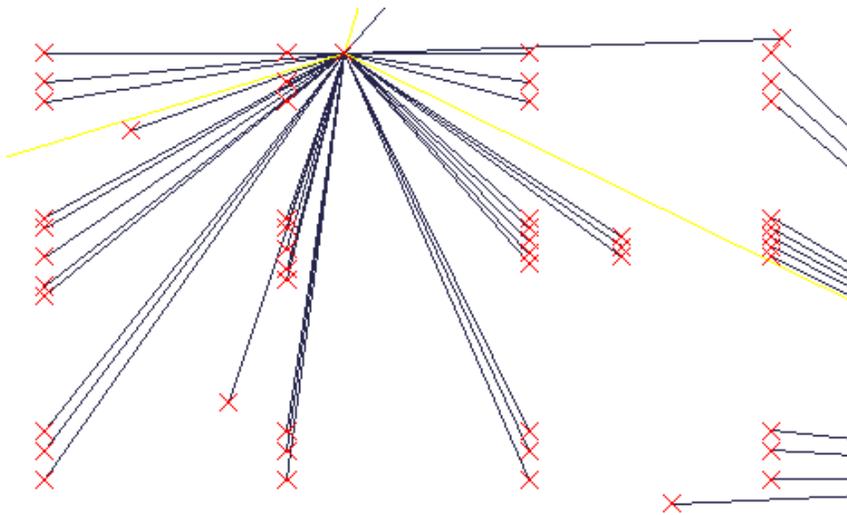


**Survey data imported from a field book**
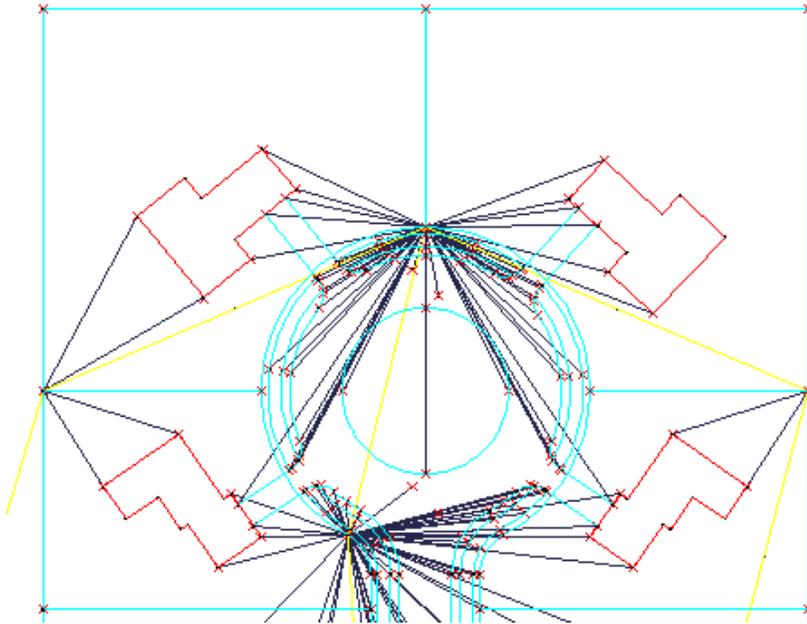
**Buildings, lots, and roadway figures created from the survey data**

| For information about…. | See... |
| --- | --- |
| Working with Survey | <span style="color:blue">Understanding Survey</span> (page 187) |
| Workflow |  Survey Workflow |
| Tutorials |  Survey Tutorials |
| Best Practices |  Survey Data |

# Object and Label Styles

The object styles in AutoCAD Civil have general attributes, such as object color, visibility of components, linetypes, and fill patterns.

Similarly, label styles work with text format, data content, location, and graphic elements, such as leader lines and bounding boxes.

Within your design process, object and label styles should be created with specific purposes in mind, such as representing objects at different approval stages, or displaying the right information for different types of users. Before you create object styles, experiment with editing styles for different objects to learn the available controls.

Style components vary according to object type, but most object and label styles are accessible in the same two ways:

■ In Toolspace, on the Prospector tab, right-click an object and click Properties to see the object style. Some objects also show the label style here. Click the list button next to the style name and select Edit Current Selection to review or edit the style components.

■ In Toolspace, on the Settings tab, expand the object collection to see all the styles for that object, organized by types. Right-click any style name and click Edit to review or edit the style components.

For objects, you can add labels by using the Annotate tab to access the Add Labels dialog box or feature-specific label menus.

You can create sets of label styles for alignments, profiles, and sections in order to manage multiple labels easily. After the set is defined, it can be applied to or removed from an object in a single operation.
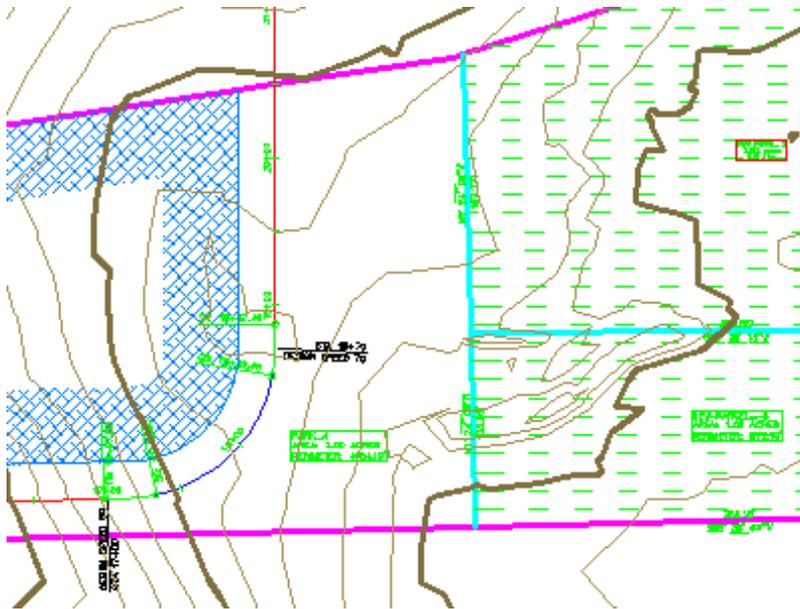
**Object styles for a surface, parcels, and an alignment, showing stylization**
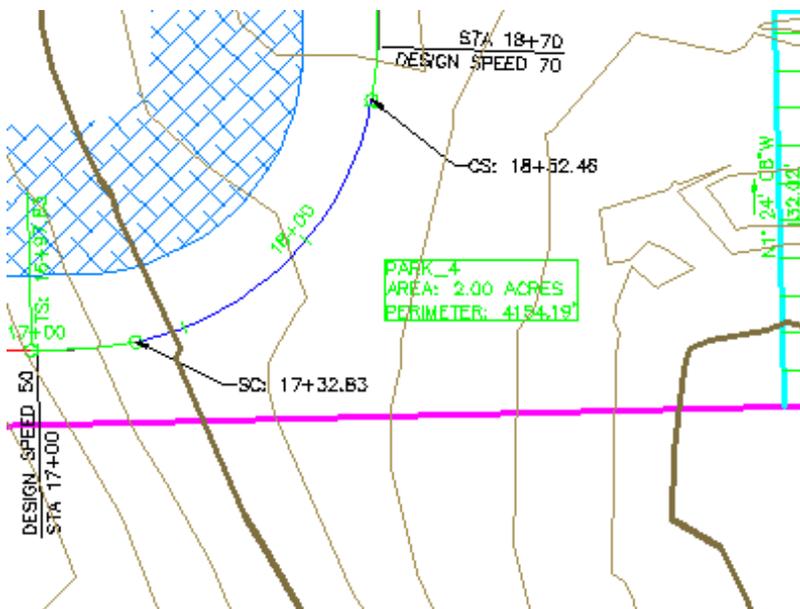


**Label styles, including two alignment labels in a dragged state**

| For information about…. | See... |
| --- | --- |
| Working with Styles | Understanding Objects and Styles (page 51) |
| Tutorials | Tutorial: Working with Label Styles |
| Best Practices | Using Styles |

# Label Objects

You manage label objects in AutoCAD Civil with standard AutoCAD commands as well as AutoCAD Civil commands. You can use the AutoCAD Properties palette to edit multiple labels in a single operation. For example, you can use the Properties palette to access the Text Component Editor in order to simultaneously edit text content for a group of any number of labels.
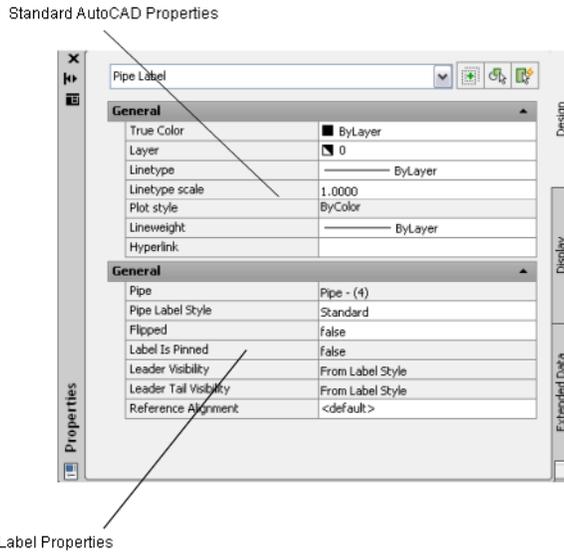


**Label Properties controlled with AutoCAD Properties palette**

If you are editing a drawing that includes dozens of pipe network parts with corresponding labels, you can edit the text, change the appearance, or reset the location of all of the labels at the same time by using multiple selection. It is important to remember that you can also select subentities of group type labels with the Ctrl-click selection method in order to edit individual labels one at a time.

As independent objects, label objects are not "subentities" of a parent object. They reside on their own layer, the default of which is controlled using drawing settings. You can edit labels using object label contextual menus. Right-click a label to display the context menu which displays options for editing properties or label behavior.
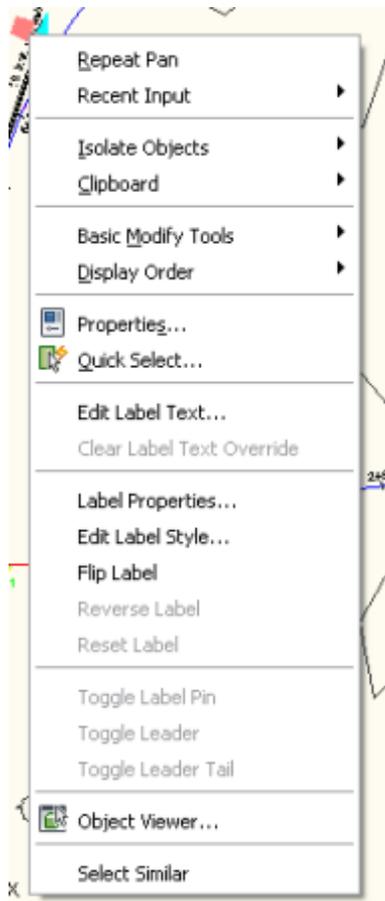
**Label object context menu**

| For information about…. | |
| --- | --- |
| Working with Label Objects | Understanding Labels (page 1466) and Label Objects (page 1463) |
| Workflow | Labels and Tables Workflow |
| Tutorials | Labels and Tables Tutorials |
| Best Practices | Label Styles and Labeling External References |

# Visualization

You can use the visualization tools in AutoCAD Civil to create enhanced design drawings using model rendering materials that correspond to specific real world materials in a design. AutoCAD render materials simulate the characteristics of materials such as asphalt, concrete, and gravel.

To visualize a material with a render material, the material must be included in the current drawing. You can easily add materials to a drawing from the tool palette or by dragging materials used in one drawing into another drawing. You can also create a new render material, and save it to use as part of a drawing template or copy it into other drawings.

In AutoCAD Civil, you can apply render materials to features, such as surfaces and surface masks, corridor surfaces and corridor surface boundaries, and pipes and structures.
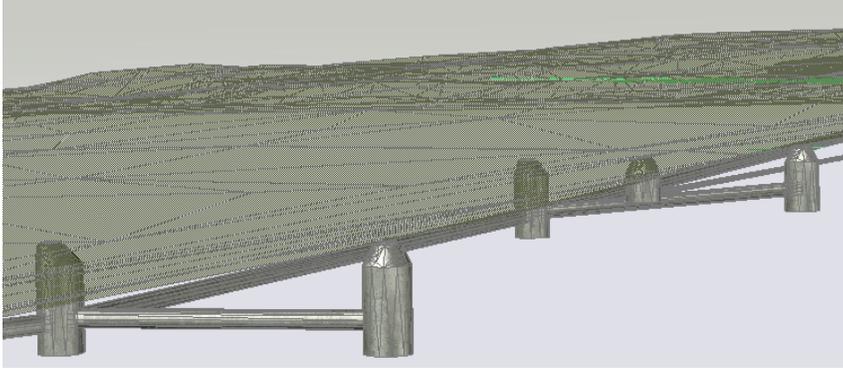
**View using styles such as 2D wireframe and 3D realistic**
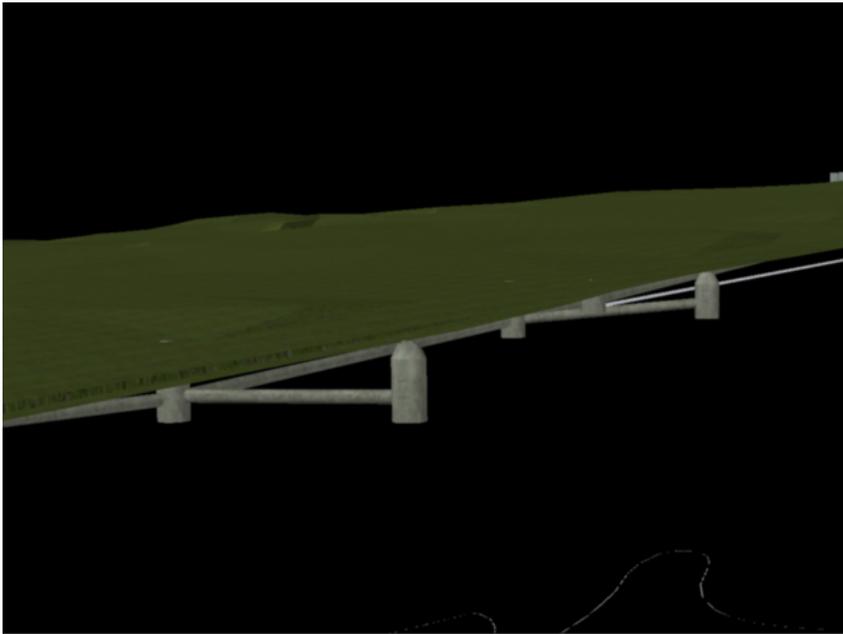


**Rendered image of a surface object simulated with grass and the underlying concrete drainage pipes and catch basins**

| For information about…. | See… |
|---|---|
| Working with Visualization | Object Rendering (page 1577) |

# Plan Production Tools

Use the Create View Frames and Create Sheets wizards to create sheets that automatically display segments of alignments and profiles in your plans. You can create view frame groups that automatically capture predefined areas along an alignment.

Instead of needing to manually create many viewports on layouts to show segments of alignments, you can create view frames that automatically capture predefined areas along an alignment. View frames are rectangular areas along an alignment that represent what is displayed in the associated viewports on the layouts (sheets) to be created. This automation saves you from making many manual changes when your design data changes.
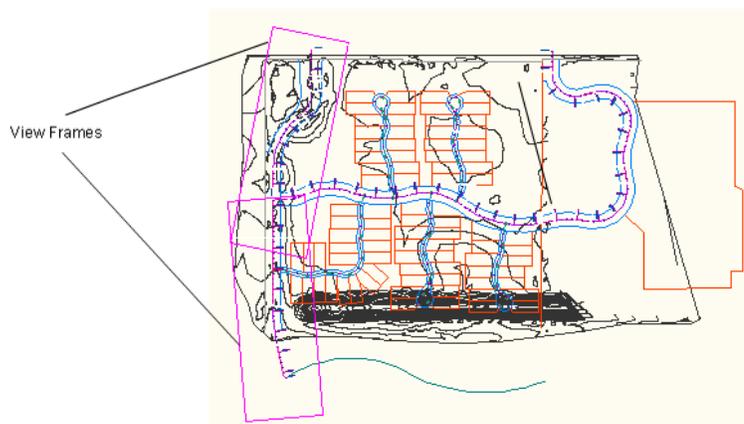


**View frames represent the alignment data that will be displayed in layout sheets**

The plan production tools simplify the process of preparing sheets from your design drawings. After you select an alignment in your drawing, in just a few seconds you can create sheets that automatically display the desired data.

| For information about…. | See… |
|---|---|
| Working with Plan Production | Plan Production Tools (page 1691) and Plan Production Tools Dialog Boxes (page 2081) |
| Workflow | Plan Production Tools Workflow |
| Tutorials | Plan Production Tutorials |
| Best Practices | Preparing Final Plan Drawings |

# Understanding Objects and Styles

# 3

In AutoCAD Civil, objects are the basic building blocks that enable you to create design drawings.

The underlying program code for AutoCAD Civil uses an object-oriented architecture. As a result, design entities in the drawing, such as points and surfaces, are intelligent objects that maintain relationships with other objects. For example, if a horizontal alignment is modified, any profiles and sections based on that alignment are automatically changed.

The primary AutoCAD Civil object types that enable you to create civil engineering design components, and the icons that represent them, are as follows:

| | | | |
|---|---|---|---|
|  | Point |  | Point Group |
|  | Surface |  | Parcel |
|  | Grading |  | Alignment |
|  | Profile |  | Profile View |
|  | Sample Line |  | Feature Line |
|  | Section |  | Section View |
|  | Mass Haul Line |  | Mass Haul View |
|  | Pipe Network |  | Pipe |
|  | Structure |  | Pipe Interference Check |
|  | Corridor |  | Assembly |
|  | Intersection |  | Subassembly |
|  | Site |  | Building Site |

| | | | |
|---|---|---|---|
| Survey Network | | Survey Figure | |
| Match Line | | View Frame | |

These types of objects are sometimes referred to as graphical objects or drawing objects because when you use them, a graphical object or shape, for example, a pipe network, surface, or corridor, is inserted into the drawing.

Use the Prospector and Settings tabs in Toolspace to access detailed object properties and settings. For more information, see The Toolspace Window (page 80).

For information on basic object behavior, see the AutoCAD Help.

# Object Relationships

AutoCAD Civil objects automate the design process by interacting with other AutoCAD Civil objects.

A design team typically spends many hours ensuring that revisions are transferred correctly between surfaces, alignments, profiles, sections, and other design data. Redrafting, relabeling, and checking the work can be time-consuming tasks. AutoCAD Civil eliminates the need for most of these tasks by introducing dynamic links between design objects. This system of links and dependencies derives from the object model within the application design.

The following diagram illustrates how AutoCAD Civil data objects are related to each other.
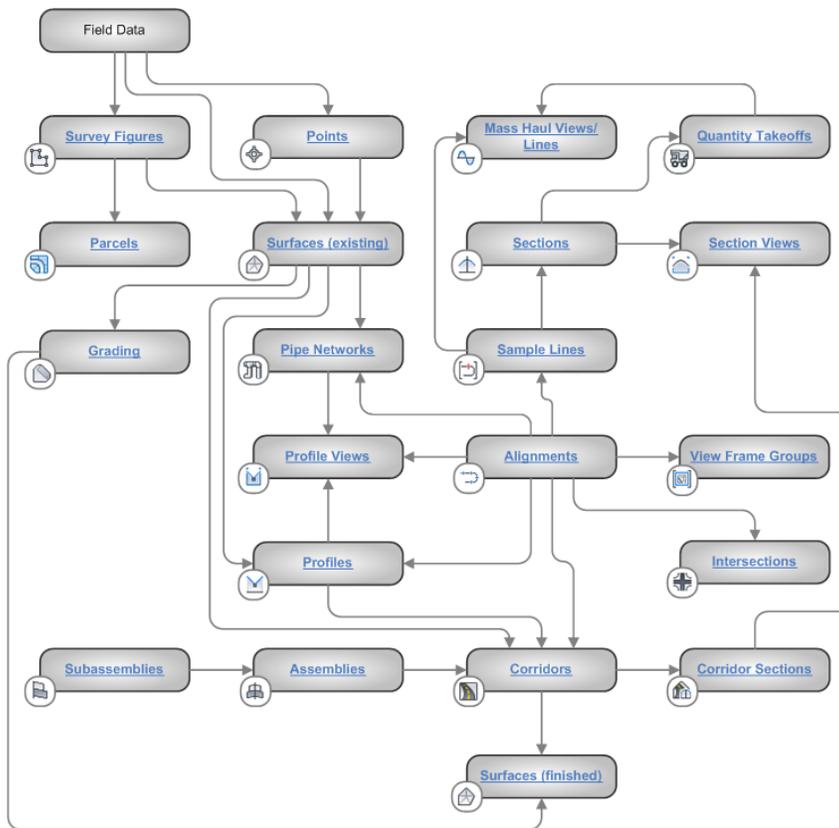


Field data creates a set of points that are used to generate an existing ground surface. This surface is referenced by other objects as indicated by the arrows.

Parcels, existing ground surfaces, pipe networks, and gradings can be created independently, or from data sources not shown in the illustration. Such objects are usually linked to other objects during the design process, if not at the beginning.

The object type with the most complex set of relationships is the corridor, as it requires data from a surface, alignment, profile, and assembly (and typically multiple subassemblies).

Changes to any object flow downward along the arrows to dependent objects, with predictable results. For example, if you correct the elevations of an existing ground surface, updates flow to any related grading objects, pipe networks, corridors, and profiles. As a result, all values represented in labels and tables are also updated.

In the design process, after you create an alignment you can create many profiles and sections. But the display of these in profile views and section views is optional and apart from the flow of data required to create the final surface. Similarly, the data from objects such as parcels and alignments can be output to a table or report if desired.

In the object model, changes in one object can be passed on automatically to associated objects where desired. For example, if you redesign an alignment curve, any grading using that alignment as a baseline can be modified accordingly. In addition, all related stationing, labels, and other alignment-specific data is updated.

The following table shows which objects can be updated when you edit each type of object:

| When you edit this object type... | These objects may be updated... |
| --- | --- |
| Points | Surfaces |
| Surfaces | Grading, Profiles, Pipe Networks, Corridors |
| Parcels | Grading, Corridors |
| Alignments | Grading, Parcels, Corridors, Profiles, Sections, Pipe Networks, Intersections |
| Profiles | Intersections |
| Grading | Surfaces, Corridors |
| Subassemblies | Assemblies, Corridors |
| Assemblies | Corridors |
| Pipe Networks | Surfaces, Alignments |
| Feature Lines | Grading |
| Sample Lines | Sections, Mass Haul diagrams |

**Objects, Styles, and Labels**

A relationship exists between drawing objects, the styles that control their display, and the labels that control their annotation. These styles and labels are also managed as objects within AutoCAD Civil.

# Object Styles

Styles control the display and design characteristics of drawing objects.

You can use styles to efficiently manage object appearance. When you create a new object, you can apply a predefined style for its display. Later, you can apply a different style. Also, you can create new styles to suit the needs of different users and different project stages. If you change a style definition, the changes are applied automatically to all objects using that style.

Styles for each object type are managed on the Toolspace Settings tab. The General collection contains styles that can be used by more than one object type (called Multipurpose styles) as well as shared label styles. All AutoCAD Civil objects have a Basic style that can be used as is, or as the basis for building new styles. If you want to customize some attributes of a style, you can create a new style, or make changes to an existing style and save it with a new name. Groups of styles can be collected and saved as a drawing template (*.dwt*) file. All drawings created from a specific *.dwt* will share the same styles. The controls for creating styles are standardized as much as possible across all features to make the process easier.

Styles Best Practices

## Working with Styles

Use the shortcut menus to make changes in object styles or object style collections.

Right-click an object style collection in the Toolspace Settings tab to create a new style.

When you right-click an existing style, you can edit, copy or delete the style. You cannot delete a style that is referenced in the drawing.

For information on creating, editing, copying, and deleting styles for a specific feature, see the styles topics for that object type:

| Object Type | See ... |
| --- | --- |
| Points | Point Styles (page 419) |
| Surfaces | Surface Styles and Visualization (page 692) |
| Parcels | Parcel Styles (page 828) |
| Profiles/Profile Views | Styles and Display of Profiles and Profile Views (page 1027) |
| Alignments | Alignment Styles (page 885) |
| Corridors | Corridor Styles and Display (page 1329) |
| Grading | Using Grading Styles (page 737) |
| Subassemblies | Subassembly Styles (page 1395) |
| Pipe Networks | Pipe Network Styles and Display (page 1187) |
| Sample Lines/Sections/Section Views | Sample Line, Section, and Section View Styles and Display (page 1130) |
| Survey | Survey Styles and Display (page 195) |
| View Frames | View Frame Styles and Display (page 1695) |
| Match Lines | Match Lines Styles and Display (page 1697) |

## Multipurpose Styles

Use the Multipurpose Styles collection to create specific styles that can be used by more than one object type.

For example, Slope Pattern style can be shared by gradings and corridors and Marker Styles can be shared by points and survey components.

For more information, see General Collection (Settings Tree) (page 94).

## Label Styles

Label Styles are used to control and manage the display of labels and expressions for a class of objects.

Labels are associated with many objects, and their content is updated whenever the object itself is changed. Labels are also controlled by styles. You can modify the label styles in the same way that you modify the object styles: right-click a style name on the Toolspace Settings tab, and then click Edit.
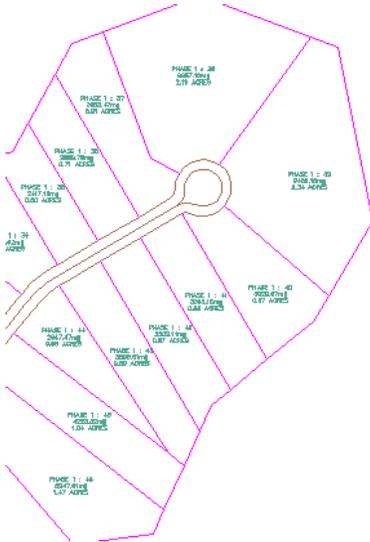


**Parcel area labels appear in the drawing as they appear in the preview of the Label Style Composer**

You can create and save label sets for alignments, profiles, and sections, which allows you to apply multiple label types in one operation. For example, an alignment label set could include labels for major stations, minor stations, and geometry points.

Labels can include text, blocks, lines, ticks, and leaders. You can create labels and preview their appearance in the Label Style Composer dialog box, as shown in the following illustration:
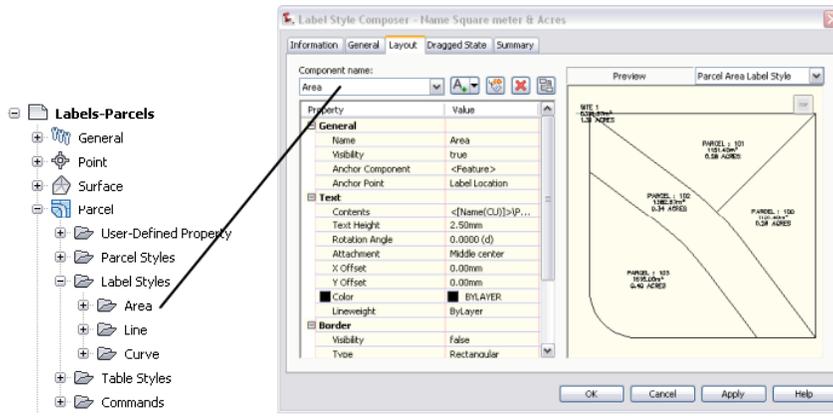
**Previewing customized label style for parcel area**

For more information about the Label Styles collections, see The Label Styles Collections (Settings Tree) (page 95).

The General Label Styles are used by lines, curves, feature lines, and corridors. This collection also contains Note label styles, which are not specific to an object.

For general information about labels, see Label Objects (page 1463).

For information about managing label styles, see Creating and Editing Label Styles (page 1477).

## Table Styles

Table Styles are used to control and manage the display for the tables associated with a class of objects.

AutoCAD Civil provides automated data tables for points, surfaces, parcels, alignments, and quantity takeoff. As shown in the following illustration, these tables provide a concise display of object data as an alternative to using object labels. The table styles control the data properties and the displayed components of the table.



**Example data table for parcels**

Data properties include the data format, order of columns, text style, and whether the table title and column headers are repeated if the table is split. Display components include the borders, separators, fill, and text. You can control the visibility, color, linetype, and scale of each component.

For information on the Table Styles collections, see The Table Styles Collections (Settings Tree) (page 96).

For general information about tables, see Tables (page 1555).

## Changing Common Settings in Styles

The Styles dialog boxes for all object types include the Information (page 1801), Display (page 1801), and Summary (page 1803) tabs. Other tabs are provided to record additional data specific to the object type.

**To change Style settings**

1 On the Settings tab in Toolspace, right-click an existing object style and click Edit.

2 Review and edit settings on the various tabs of the Style dialog box.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click object style ➤ Edit

Dialog Box

Styles (page 1801)

# Object Properties

AutoCAD Civil object properties can be modified using the civil object properties dialog boxes, the Properties palette, and the Quick Properties palette.

### Civil Object Properties

Each AutoCAD Civil object has its own dialog box which contains most properties relevant to that object.

Use these dialog boxes as the primary interface for adjusting AutoCAD Civil object properties.

For example, use the Alignment Properties dialog box to adjust properties for a selected alignment object in a drawing.

### Properties Palette

The Properties palette can be used as an alternate method of editing AutoCAD Civil object properties.

In AutoCAD Civil 2010, more civil object properties are exposed in the Properties palette than in previous releases. Properties are now organized into four categories: Information, General, Data, and Geometry.

By exposing these properties in the Properties palette, they are now also available to be used with the QSELECT command.

For more information, see *Control the Properties of Objects* in the AutoCAD Help.

### Quick Properties Palette

For each civil object, the name, description, style, and layer are available for editing in the Quick Properties palette. For more information, see *Display and Change the Properties of Objects* in the AutoCAD Help.

Also, you can easily customize the quick properties for any object in the Customize User Interface (CUI) editor. For more information, see *Quick Properties* in the AutoCAD *Customization Guide*.

### Matching Object Properties

The Match Properties command can be used with AutoCAD Civil objects. When you use the Match Properties command between civil objects, the style and object display properties of the selected source object are applied to the selected destination objects. For more information, see *Copy Properties Between Objects* in the AutoCAD Help.

**To access comprehensive AutoCAD Civil object properties**

■ Select a AutoCAD Civil object and click <Object Name> Properties on the Modify panel of the contextual ribbon tab.

OR

■ Right-click an AutoCAD Civil object and click <Object Name> Properties in the shortcut menu.

**To access AutoCAD Civil object properties in the Properties Palette**

■ Select an AutoCAD Civil object and click Properties on the General Tools panel of the contextual ribbon tab.
OR

■ Right-click an AutoCAD Civil object and click Properties in the shortcut menu.

**To access AutoCAD Civil object properties in the Quick Properties Palette**

1 Enable the Quick Properties mode by entering QPMODE at the command line and entering 1. Or click the Quick Properties button on the Drawing Status Bar.

2 Select an AutoCAD Civil object in the drawing.

3 In the Quick Properties Palette, edit the desired properties.

# Layers

Each object in AutoCAD Civil has a *base layer* on which the object physically resides, and *component layers* that control the display of object components, such as surface triangles or contours.

The base layer is defined in the Drawing Settings or at creation time. The component layers are defined in the object styles.

**Object Base Layer**

You specify the default base layers for objects on the Object Layers tab of the Drawing Settings dialog box.
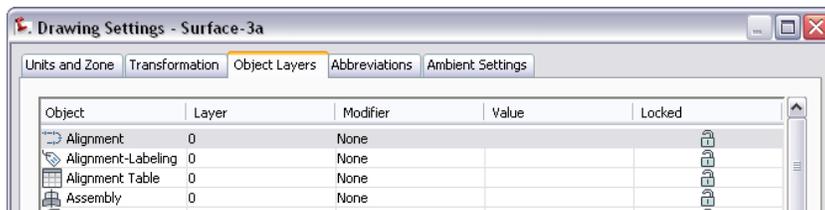


When you create an object, you can accept the default or specify a new layer. The base layers are used only for the main object types, such as surfaces and alignments.

Because the object physically resides on the base layer, you can control object visibility by changing the layer state. For example, if you turn off the base layer for surfaces, all surfaces are turned off in the drawing.

**TIP** To append an object name to the layer name for that object, add an asterisk (*) as a prefix or suffix to the base layer name.

For more information, see Object Layers Tab (Drawing Settings Dialog Box) (page 1854).

**Object Component Layers**

You specify component layers on the Display tab of the <object> Style dialog box. The following example shows layers used for alignment components:
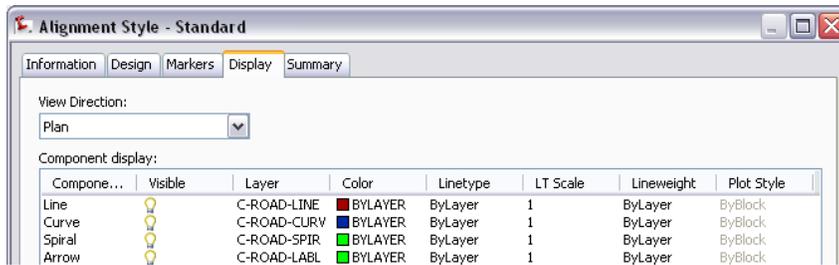
The component layers, such as C-ROAD-LINE, allow you to work with objects as if parts of them are located on different layers. For example, to turn off only the alignment lines, you can turn off the C-ROAD-LINE layer in the drawing.

Objects are physically created on the base layer, but the display of components is governed by the associated component layers. Each object component uses the visibility settings for the component layer (such as on/off), as well as the color, linetype, lineweight, and plot style for the layer if the various Component Display settings are set to ByLayer.

**NOTE** Layer 0 has a special meaning in the Component Display settings. When the layer is set to 0, the object base layer is used for that component. For example, if the alignment base layer is C-ROAD, and the layer for the Line component is set to 0, then the alignment lines behave as if they are located on layer C-ROAD (not layer 0).

For more information, see Display Tab (Style Dialog Box) (page 1801).

**Setting Up Styles Using the ByLayer Setting or Specific Settings**

If you are accustomed to controlling objects by manipulating layers, you may want to set the various object style Component Display settings to ByLayer. Then you can simply change the layer settings using the AutoCAD Layer command rather than editing separate styles to change the color, linetype, lineweight, or plot style of an object component.

**TIP** Using the ByLayer settings provides more control over object display to people viewing AutoCAD Civil drawings in object-enabled AutoCAD.

However, if you want the object to always have the same settings regardless of component layer settings, you can set specific Component Display settings.

For more information, see Using ByLayer and ByBlock to Assign Display Values (page 1803).

# Naming Constraints for Objects and Styles

Object and style names restrict the use of certain characters.

When naming an object or style, do not use the following characters:

- < (less than)
- > (greater than)
- / (forward slash)
- \ (backward slash)
- + (plus sign)
- " (double quotes)
- : (colon)
- ; (semi colon)

- ■ = (equals)
- ■ | (vertical line)
- ■ , (comma)
- ■ * (asterisk)
- ■ ? (question mark)
- ■ ' (single quote)

# Understanding Settings

# 4

AutoCAD Civil has drawing, object, and command settings. All three levels of settings in AutoCAD Civil are saved with the drawing, and they can be saved to a drawing template.

Settings in AutoCAD Civil provide many preset values, ranging from values, such as drawing units, scale, and coordinate system, to optional defaults, such as the layers that the different objects are created on, and the use of tooltips.You can access the setting dialog box by right-clicking the appropriate collection on the Toolspace Settings tab, and then clicking Edit Feature Settings.

**You can work with three levels of settings:**

■ Drawing settings establish values for the whole drawing. If you are creating a drawing template, ensure that these are set correctly.

■ Feature settings control behavior for a particular feature, such as Parcels or Grading.

■ Command settings apply to individual commands within a feature, such as the CreateParcelByLayout command within the Parcels feature.

Each lower level object in the settings hierarchy can either inherit or override settings in the level above it. The following illustration shows an override set for area units at the Parcels feature level.
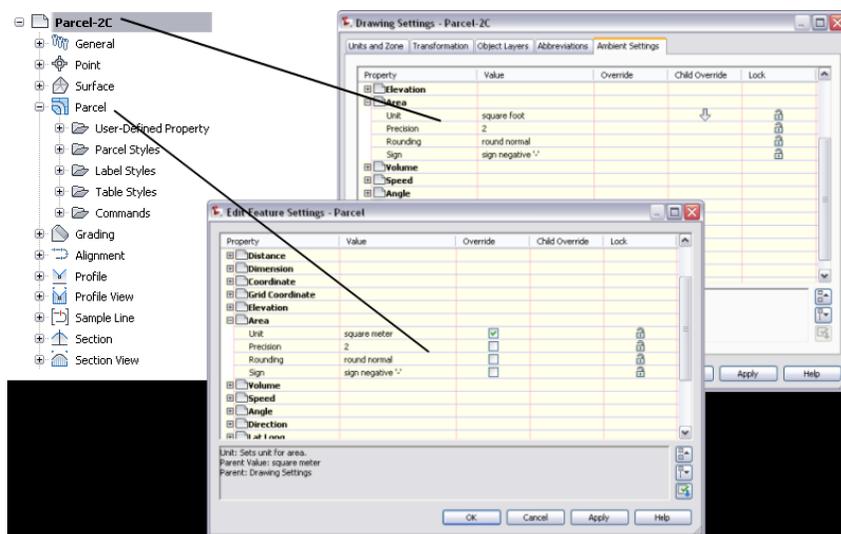


**Feature settings can override drawing settings**

The arrow in the Child Override column of the Drawing Settings dialog box (upper drawing) indicates that an override has been set at a lower level. The check mark in the Override column in the Parcel Settings dialog box (lower drawing) indicates that the value set in this dialog box overrides the setting at a higher level.

At the drawing level you can cancel an override by clicking it. You can prevent overrides by locking a setting.

# Working with the Standard Settings Dialog Box Controls

Use a standard dialog box, accessed at three different levels, to edit settings.

You modify each level of settings at a different location in the Settings tree, as shown in the following illustration:
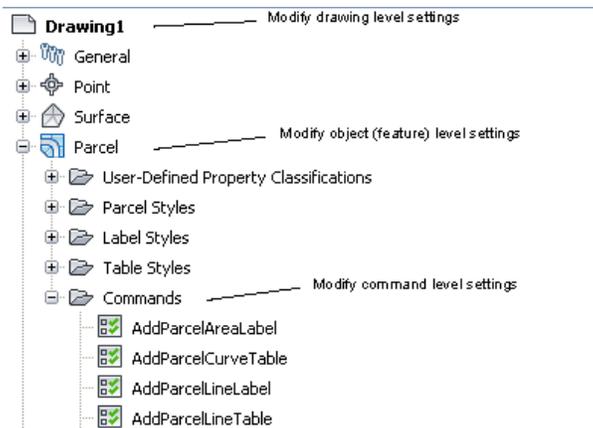


**Drawing-Level Settings**

There are two types of drawing-level settings:

■ Drawing-wide settings, which include units and zone, transformation settings, abbreviations, and object layers.

■ Ambient settings, which affect a variety of AutoCAD Civil behaviors. You can change ambient settings at the drawing level, and you can also override ambient setting values at either the object level or the command level.

You access drawing-level settings by right-clicking the drawing name in the Settings tree and clicking Edit Drawing Settings.

At the drawing level, the Edit Settings dialog box contains only the ambient settings for the drawing:



**Object(Feature)-Level Settings**

Use the object collection level in the Settings tree to control all settings that pertain to the object type. Change the object settings at this level in the tree, and specify overrides for the drawing ambient settings.

Access the object- or feature-level settings by right-clicking the object collection in the Settings tree and clicking Edit Feature Settings.

At the object level, this dialog box also contains object-specific settings, such as default styles, which are listed below the General settings:

| Property | Value | Override | Child Override | Lock |
|---|---|---|---|---|
| ⊞ 🗋 General | | | | |
| ⊞ 🗋 Labeling | | | | |
| ⊞ 🖫 Default Styles | | | | |
| ⊞ 🗋 Unitless | | | | |
| ⊞ 🗋 Distance | | | | |

**Command-Level Settings**

Use the Commands collection level in the Settings tree to override both the object-level settings and drawing ambient settings on a command-by-command basis. You can also specify command-specific settings.

Access the command-level settings by expanding the Commands collection for an object type in the Settings tree, right-clicking the command, and clicking Edit Command Settings.

At the command level, this dialog box also contains command-specific settings, such as the Parcels and Alignments settings that are used by the CreateSite command.

| Property | Value | Override | Chi |
|---|---|---|---|
| ⊞ 🗋 General | | | |
| ⊞ 🗋 Labeling | | | |
| ⊞ 🖫 Default Styles | | | |
| ⊞ 🗹 Parcels | | | |
| ⊞ 🗹 Alignment | | | |
| ⊞ 🗹 FeatureLine | | | |
| ⊞ 🗋 Unitless | | | |

The following summarizes the controls in the Edit Settings dialog box:

■ Use the + or – boxes to display or hide the settings in the category.

■ Use the Value column to specify a value for a setting.

■ The Override column indicates that the default value has been changed for the setting.

■ The Child Override column indicates whether the setting has been overridden at a lower level in the Settings tree.

■ Use the Lock column to control whether a setting can be changed at a lower level in the Settings tree.

For more information, see .

# Specifying Drawing Settings

Drawing Settings are the most generic settings. All commands use the drawing settings unless there is a specific override at the feature or command level.

# Specifying Units and Zone Settings

Using the Units and Zone tab of the Drawing Settings dialog box, you can select linear and angular units, a coordinate system, and scale for the drawing.

**Coordinate Systems**

Coordinates in AutoCAD Civil are expressed using northings and eastings.

The AutoCAD Civil coordinate system is always relative to the AutoCAD World Coordinate System (WCS). Northing is equivalent to AutoCAD *Y* and Easting is equivalent to AutoCAD *X*.

AutoCAD Civil object data is always presented in AutoCAD World Coordinate System coordinates. For example, the alignment data in the Alignment Entities vista in the Panorama window is listed in world coordinates. In addition, the AutoCAD Civil transparent commands are based on the AutoCAD World Coordinate System. However, when you create an object and have an AutoCAD User Coordinate System (UCS) defined, the UCS is honored when you specify locations, which means that an insertion point or a coordinate is interpreted relative to the UCS.

AutoCAD Civil objects are not aligned with the UCS. For example, labels can be oriented relative either to the current view, the World Coordinate System, or the labeled object, depending on the Orientation Reference setting, but they cannot be oriented to the UCS.

Similarly, objects, such as profiles, sections, and tables, cannot be oriented to the current UCS.

For more information about AutoCAD World Coordinate Systems and AutoCAD User Coordinate Systems, see the AutoCAD Help.

**To specify Units and Zone settings**

1 In Toolspace, on the Settings tab, right-click the drawing name at the top of the collection, and click Edit Drawing Settings.

2 In the Drawing Settings dialog box, click the Units and Zone tab (page 1851).

3 Under Drawing Units, select either Feet or Meters.

4 Under Angular Units, select an angle format.

5 Specify one of the following for the Imperial to Metric Conversion.
   ■ International Foot (1 foot = 0.3048 Meters)
   ■ US Survey Foot (39.37 Inches per Meter)

6 Select the Scale Objects Inserted From Other Drawings check box to scale objects inserted from another drawing to match drawing units in the current drawing.

7 Select the Set AutoCAD Variables To Match check box to synchronize AutoCAD settings with AutoCAD Civil settings. If there are no equivalent AutoCAD settings, a message is displayed asking if you want to match as closely as possible. The AutoCAD settings that are synchronized to theAutoCAD Civil settings include the AUNITS, DIMAUNIT, INSUNITS, and MEASUREMENT AutoCAD system variables (sysvars).

8 Under Scale, select the intended plot scale in imperial or metric units.

9 Under Custom Scale select the plotted size of various annotation-related components, such as label text, ticks, and band heights. If you change the scale, all annotation objects adjust accordingly.

10 Under Zone, select a category from the Categories list.

   Categories include Lat/Longs, US states, and so on. When you select a category, the Available Coordinate Systems list displays all the defined zones in that category.

   **TIP** If you know the coordinate system (CS) code for a zone, you can enter it in the CS Code text box and click OK to select the zone.

11 Click OK.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: right-click the drawing name  ➤ Edit Drawing Settings

Dialog Box

Drawing Settings (page 1851)

# Specifying Transformation Settings

Using the Transformation tab of the Drawing Settings dialog box, you can relate the local northing and local easting coordinates of your drawing to the grid northing and grid easting coordinates for the current zone.

The zone transformation settings do the following:

■ Relate local coordinates to grid coordinates by transforming distances measured on the Earth (or geoid) to distances on an ellipsoid.

■ Relate distances on the ellipsoid to the flat plane (projection) of the current zone's grid coordinate system.

These transformations are accomplished through the use of scaling factors. First, a sea level scale factor is applied to the local values measured on the geoid, and then a grid scale factor is applied, which relates the ellipsoid values to the grid projection. The scaling factors can be defined in two ways:

■ The sea level scale factor relates the distances on the geoid to the distances on the ellipsoid.

■ The grid scale factor relates the distances on the ellipsoid to the distances on the grid projection.

You must also specify reference points in establishing transformation settings.

These reference points are the two points that tie the local and grid coordinates together. The reference points can be defined in two ways:

■ By the grid and local coordinates of two known reference points in your drawing.

■ By the grid and local coordinates of one known point and a known rotation to grid north.

**To specify transformation settings for a drawing**

1   In Toolspace, on the Settings tab, right-click the drawing name  at the top of the collection, and click Edit Drawing Settings.

2   In the Drawing Settings dialog box, click the Transformation tab (page 1852).

   **IMPORTANT** You must specify the coordinate system on the Units and Zone tab (page 63) before you can specify settings on the Transformation tab.

3   Select the Apply Transform Settings check box to activate all the controls in the dialog box. Enter values as specified in steps 4-10. If you clear this check box, then controls in the dialog box are disabled and the AutoCAD X and Y coordinates match the coordinates in the specified zone.

4   Select Apply Sea Level Scale Factor.

> **TIP** If you know the combined scale factor, then you can clear the Apply Sea Level Scale Factor check box, select User Defined as the Grid Scale Factor, and enter a combined scale factor in the Grid Scale Factor box. The combined scale factor is the combination of the scale factor for converting local to sea level and the scale factor for converting sea level to grid.

**5** Enter a default Elevation, such as the average elevation of your project site from sea level.

**6** Optionally, change the Spheroid Radius, which is the radius of a mathematical figure close to the shape of the Earth at sea level, approximately 6,370 km. The value shown in this box is initially derived from the current zone's ellipsoid and can be changed if local observations differ. In most cases, the default value shown is the accepted value.

**7** For the Grid Scale Factor, select a method for Computation. For more information, see Transformation tab (page 1852).

**8** Specify the Reference Point values. The reference point could be a benchmark that was used in a survey. It can be any point for which you know both the local coordinates and the grid coordinates. To specify the Reference Point values, do one of the following:

- Click the Select Point button and then select a point in the drawing, or use Transparent commands.

- Enter a point number.

- Enter the point's Grid Northing and Grid Easting values.

**9** To define the rotation angle for the transformation, do one of the following:
- Click the Select Point button and then select a point in the drawing.

- Enter a point number.

- Enter the point's Grid Northing and Grid Easting values.

**10** Specify the grid rotation angle.

**11** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click the drawing name  ➤ Edit Drawing Settings

Dialog Box

Drawing Settings (page 1851)

## Specifying Object Layer Settings

Use the Object Layers tab of the Drawing Settings dialog box to preset default object layers.

For more information about object layers, see Layers (page 58).

**To specify object layer settings for a drawing**

**1** In Toolspace, on the Settings tab, right-click the drawing name  at the top of the collection, and click Edit Drawing Settings.

**2** In the Drawing Settings dialog box, click the Object Layers tab (page 1854).

**3** For each object type, click in the Layer column to display the Select Layer dialog box.

**4** In the Select Layer (page 1986) dialog box, select the layer for that object type and click OK.

**5** To add a modifier to the layer name, select Prefix or Suffix in the Modifier column and then enter the text string for the modifier in the Value column.

> **TIP** To include the object name in the layer prefix or suffix, enter an asterisk in the Value column.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click the drawing name  ➤ Edit Drawing Settings

Dialog Box

Drawing Settings (page 1851)

## Specifying Abbreviation Settings

Use the Abbreviations tab of the Drawing Settings dialog box to change the abbreviations used for the labels.

The Abbreviations tab controls various default abbreviations used in drawing labels and reports. For example, the default abbreviation for Spiral-Tangent Intersect is TS, meaning any spiral-tangent intersections in a drawing are labeled TS. If you need spiral-tangent intersection labels to be more detailed or different, you can change the abbreviation by clicking in the value column and changing the TS value.

In the Alignment Geometry Point Entity Data section the values also contain format strings that describe entity properties that will display in the label. For more information see, Property Field Modifiers (page 1509).

**To specify abbreviation settings for a drawing**

**1** In Toolspace, on the Settings tab, right-click the drawing name  at the top of the collection, and click Edit Drawing Settings.

**2** In the Drawing Settings dialog box, click the Abbreviations tab (page 1855).

**3** Specify the abbreviations for the properties by modifying the entries in the Value column.

> **NOTE** For individual abbreviation definitions, see the Glossary (page 2467).

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click the drawing name  ➤ Edit Drawing Settings

Dialog Box

Drawing Settings (page 1851)

## Specifying Ambient Settings

Using the Ambient Settings tab of the Drawing Settings dialog box, you can specify default ambient (background) settings for a variety of features, including some that affect units of measurement.

These settings control parameters such as angle, area, direction, elevation, and so on. There are also settings that affect the Event Viewer, tooltips, and how curb returns are created for intersections.

**To specify ambient settings for a drawing**

1  In Toolspace, on the Settings tab, right-click the drawing name ▭ at the top of the collection, and click Edit Drawing Settings.

2  In the Drawing Settings dialog box, click the Ambient Settings tab (page 1856).

3  In the Property column, select a parameter in one of the categories.

   As you select a parameter, the description in the lower part of the dialog box reflects the selection.

4  Change the settings in the Value column.

5  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click the drawing name ▭ ➤ Edit Drawing Settings

Dialog Box

Drawing Settings (page 1851)

# Specifying Feature-Level Settings

Specify object-specific settings and override drawing ambient settings at the object (feature) collection level of the Settings tree.

**To specify feature-level settings**

1  In Toolspace, on the Settings tab, right-click the object collection and click Edit Feature Settings.

2  To change an object-specific setting, do the following in the Edit Settings (page 1809) dialog box:
   ■  Expand the category that contains the setting.

   ■  Click the cell in the Value column. Enter the value or select a value from the list.

3  To override a drawing ambient setting, do the following:
   ■  Expand the category that contains the setting.

   ■  Click the cell in the Value column. Enter the value or select a value from the list.

   After you change the value, the check box in the Override column is automatically selected, indicating that the drawing ambient setting is overridden.

4  Click OK.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: right-click object collection ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings (page 1809)

# Specifying Command-Level Settings

Override the object-level settings or the drawing ambient settings for a specific command at the command level of the Settings tree.

**To specify command level settings**

1 In Toolspace, on the Settings tab, expand the Commands collection that contains the command whose settings you want to edit.

2 Right-click the command name and click Edit Command Settings.

3 To change a command-specific setting, do the following in the Edit Settings (page 1809) dialog box:
   ■ Expand the category that contains the setting.

   ■ Click the cell in the Value column. Enter the value or select a value from the list.

4 To override object-specific settings, including drawing ambient settings, do the following:
   ■ Expand the category that contains the setting.

   ■ Click the cell in the Value column. Enter the value or select a value from the list.
   After you change the value, the check box in the Override column is automatically selected, indicating that an object-level setting is overridden.

5 Click OK.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: right-click command name in object Commands collection ➤ Edit Command Settings

Dialog Box

Edit Command Settings (page 1809)

# The User Interface

# 5

The AutoCAD Civil user interface enhances the standard AutoCAD environment with additional tools for creating and managing civil design information.

Standard AutoCAD features, such as the command line and the design space work the same way in AutoCAD Civil as they do in AutoCAD.

## Ribbon

AutoCAD Civil commands and features are available from the ribbon.

## AutoCAD Civil Ribbon Overview

The AutoCAD Civil ribbon is the primary user interface for accessing commands and features.

Commands available from the ribbon are organized into tabs. Each tab is organized into a series of panels, which are labeled by task.

The ribbon is typically turned on (displayed) by default, and can be displayed or removed using the Ribbon and RibbonClose commands.

There are two basic types of ribbon tabs: static and contextual.

### Static Ribbon Tabs

The AutoCAD Civil static ribbon tabs include the Home, Insert, Annotate, Modify, Analyze, View, Output, and Manage tabs. Static ribbon tabs are always displayed when the ribbon is turned on, and they contain most of the functionality located on the menu bar drop-down menus and toolbars. Static ribbon tabs have a gray background color.
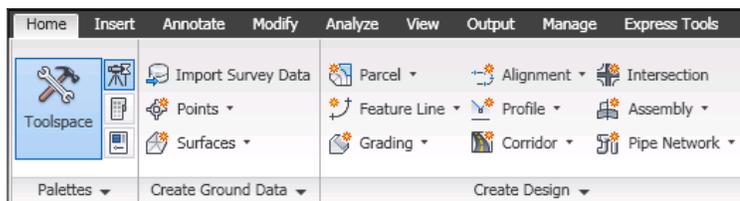
**Contextual Ribbon Tabs**

Contextual ribbon tabs are displayed automatically when you select an object or invoke an object-specific command. For example, when you select a pipe network object, the Pipe Network contextual tab is displayed.

Contextual tabs contain commands that are related to the currently selected object. Most contextual tabs can be closed simply be deselecting the object.

AutoCAD Civil contains the following contextual tabs:

- Alignment
- Assembly
- Cogo Point
- Corridor
- Feature Line
- Grading
- Intersection
- Label
- Mass Haul Line
- Mass Haul View
- Profile
- Profile View
- Parcel
- Plan Production
- Pipe Network
- Projected Object
- Sample Line
- Section
- Section Sheet
- Section View
- Subassembly
- Surface
- Survey
- Table

**Multiple Object Select Ribbon Tabs**

When multiple types of objects are selected simultaneously, the Multiple contextual tab is displayed in the ribbon. For example, if you select both a pipe network object and an alignment object, the Multiple contextual tab is displayed.

**Section Editor Ribbon Tab**

The Section Editor tab includes some ribbon behavior that is slightly different than the other types of contextual ribbon tabs just described. The Section Editor tab lets you perform a variety of corridor section view editing functions. You can display the Section Editor tab by clicking Corridor Section Editor on the Modify panel of the Corridor contextual tab, or by selecting View/Edit Corridor Section from the corridor object shortcut menu. This tab remains active until you click the Close button to remove it.

**NOTE** To learn more about basic ribbon functionality, see *The Ribbon* in the AutoCAD User's Guide Help.

# Contextual Tab Colors

The background color of a contextual tab depends on the type of task being performed.

### Object Select Contextual Tabs (Green)

Contextual tabs that are displayed when you select an object have a green background color. For example, if you select an assembly object, the Assembly contextual tab is displayed. That tab is displayed with a green background color.

The following illustration shows the assembly object select contextual tab.



### Non-object Select Contextual Tabs (Purple)

Contextual tabs that are displayed when no objects are selected have a purple background color. For example, if you click the Assembly button  on the Modify tab, the Assembly non-object select contextual tab displays, and that tab has a purple background color. This provides access to all commands related to assemblies without having to select an assembly in the drawing.

The following illustration shows an example of a contextual tab that is displayed when no objects are selected.
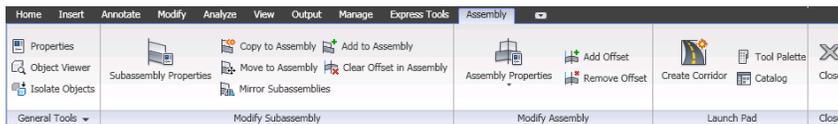


Non-object selection contextual tabs have a Close button at the end to dismiss them. If you select an object while this tab is displayed, the non-object selection tab is automatically closed and replaced with the object select contextual tab.

### Section Editor Tab (Blue)

Because its ribbon behavior is slightly different than the previous two types just described, the Section Editor tab has its own unique background color — light blue (teal).
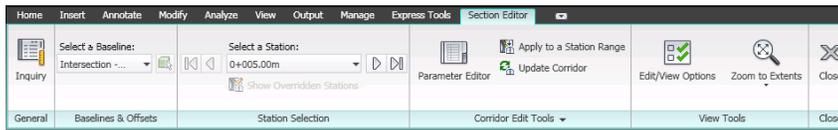
## Launch Pad Panel

AutoCAD Civil contextual ribbons include a Launch Pad panel that provides access to commands that you might use next in your workflow.

For example, when you select an alignment object, the Alignment contextual tab is displayed. The Launch Pad panel on the Alignment contextual tab provides a variety of commands that you may want to do next. For example, you may want to create a corridor, intersection or data shortcut from the alignment.

Similarly, the Launch Pad on the Corridors contextual tab provides access to the variety of commands that you may want to select while you have a corridor object selected. For example, you can access the Create Sample Line command and the Create Feature Lines From Corridors command, among others, from this Launch Pad.

## Using the Where is My Command? Tool

Use the Where is My Command? tool to quickly locate AutoCAD Civil 2009 menu commands on the AutoCAD Civil 2010 ribbon.

This tool lets you look up familiar AutoCAD Civil 2009 menu commands and quickly determine where these commands now exist on the AutoCAD Civil 2010 ribbon. Access this tool from the Help drop-down menu , or click *here*.

The Where is My Command tool? contains only AutoCAD Civil 2009 menu commands. It does not identify ribbon locations for new AutoCAD Civil 2010 commands and features that did not exist in AutoCAD Civil 2009.

### Printing

It can be useful to print out some or all pages of the Where is My Command Tool? to quickly and easily reference this information any time.

To print out all of the Where is My Command? Help topics, right-click the AutoCAD Civil 2009 Menus in the left pan Contents tab, and select Print. On the Print Topics dialog box, select Print the selected heading and all subtopics.

To print out a single topic, right-click an item in the left pan, and select Print. On the Print Topics dialog box, select Print the selected topic, or place your cursor in the right pane, and select Print from the right-click shortcut menu.

### To use the Where is My Command? tool

1   In the InfoCenter toolbar located at the top right side of the application window, click the drop-down arrow next to the Help icon  .

2   Click Where is My Command?

3   In the left pane, on the Contents tab, expand AutoCAD Civil 2009 Menus.

4   In the left pane, click a menu item. For example, click Pipes to display the choices that exist on the AutoCAD Civil 2009 Pipes menu.

**5** In the right pane, look up a menu command in the Menu column, and then use the Ribbon Location column to find out where this command now exists on the ribbon.

Some AutoCAD Civil 2009 menu commands do not exist on the ribbon, but instead are available from the command line or other access method. For those commands, the Ribbon Location column lists the appropriate access method.

## Learning to Use the Ribbon

AutoCAD Civil includes several resources for learning how to use the ribbon.

Refer to the following table for links to Help topics describing ribbon functionality.

| For information about ... | See.... |
| --- | --- |
| basic ribbon functionality | *The Ribbon* in the AutoCAD User's Guide Help (User's Guide ➤ The User Interface ➤ Tools in the Application WIndow ➤ The Ribbon) |
| customizing the ribbon | *Customize The Ribbon* in the AutoCAD User's Guide Help (User's Guide ➤ The User Interface ➤ Tools in the Application WIndow ➤ The Ribbon ➤ Customize The Ribbon) |
| learning how to use the ribbon | ■ *User Interface Overview* in the AutoCAD Civil User's Guide Help (Getting Started ➤ The User Interface ➤ User Interface Overview)<br><br>■ Tutorial: Understanding the AutoCAD Civil User Interface in AutoCAD Civil Tutorials (Getting Started Tutorials ➤ Tools in the Application WIndow ➤ Understanding the AutoCAD Civil User Interface) |
| finding command locations on the ribbon | Using the Where is My Command? Tool (page 74) (User's Guide Help ➤ Getting Started ➤ The User Interface ➤ Ribbon ➤ Using the Where Is My Command? Tool) |

## Quick Access Toolbar

The Quick Access toolbar displays frequently used tools.

You can add ribbon buttons to the Quick Access toolbar, by right-clicking the button on the ribbon and then clicking Add to Quick Access toolbar. Buttons are added to the right of the default commands on the Quick Access toolbar.

For more information on the Quick Access toolbar, see *Quick Access Toolbar* in the AutoCAD User's Guide Help.

## Menus

While the ribbon is the primary access point for AutoCAD Civil commands, many commands are also available from the application menu and from shortcut menus that are displayed when you right-click an object in Toolspace or in the drawing window.

### Application Menu

The application menu provides access to file-related commands. For example, it provides commands that let you create, open, print, export, and publish a file.

Access the application menu by clicking the application menu icon at the top left of the application window  .

The application menu also provides a search tool which you can use to search for commands. Only those commands that are available in the current workspace are shown and searched. For more information, see *The Application Menu* in the AutoCAD User's Guide Help.

### Shortcut Menus

Shortcut menus are available throughout AutoCAD Civil. To display a shortcut menu, right-click your pointing device on items in the Toolspace, Panorama, or other window, or on objects in the drawing area.

### Legacy Menu Bar

The menu bar is no longer displayed across the top of the AutoCAD Civil 2010 program window by default. You can display the menu bar by clicking the drop-down arrow on the Quick Access toolbar and selecting Show Menu Bar, or by entering **menubar** at the command line. The Quick Access toolbar is located at the top-left side of the application window by default.

---

**NOTE** The legacy menu bar contains only AutoCAD Civil 2009 commands and has not been updated with new AutoCAD Civil 2010 commands and features. Therefore, if you display the menu bar, it is important to understand that it does not provide access to all AutoCAD Civil 2010 features.

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# Workspaces

AutoCAD Civil 2010 comes with a default workspace for working with AutoCAD Civil commands. You can use this workspace as-is or modify it according to your requirements.

Workspaces are sets of user interface components, such as ribbon tabs and panels, toolbars, palettes, and menu bars, that are grouped and organized so that you can work in a custom, task-oriented drawing environment. When you select a workspace, only the user interface components specified in that workspace are displayed. You can still access other commands by entering their command names at the command line.

You can switch to another workspace by using the Workspace Switching button  , which is located in the application status bar.

Workspaces in AutoCAD Civil 2010 include:

- **Civil**: This workspace displays user interface components related to civil engineering design and survey features available in AutoCAD Civil.

- **2D Drafting & Annotation**: This workspace displays user interface components related to AutoCAD 2D drafting and annotation features available in AutoCAD Civil.

- **3D Modeling**: This workspace displays user interface components related to AutoCAD 3D modeling features available in AutoCAD Civil.

For more information about using and customizing workspaces, see the *AutoCAD User's Guide* and the *AutoCAD Customization Guide* in the AutoCAD Help.

**To set the workspace to the AutoCAD Civil default**

1 At the command line, enter **Workspace**.

**2** At the Workspace Option prompt, enter **C**.

**3** When prompted to enter the name of the workspace, enter:
**Civil**.

**To save the current menu and toolbar configuration to a new workspace**

**1** Configure your workspace. For example, you can close the Tool Palette if you do not want it to come up every time you start AutoCAD Civil.

> **NOTE** You can also use the CUI command to edit the Workspace using a dialog box interface. For more information, see *Customer User Interface Dialog Box* in the AutoCAD Command Reference.

**2** Click General menu ➤ Workspaces ➤ Save Current As.

**3** In the Save Workspace dialog box, enter a name for the new workspace. You can also select the name of an existing workspace to overwrite it with your modified configuration.

**4** Click Save.

## Quick Reference

Command Line

Workspace

# Accessing AutoCAD Civil Toolbars

AutoCAD Civil provides toolbars you can use to quickly access the transparent commands.

The two transparent command toolbars are turned on (displayed) by default when using the **Civil** workspace.

Other toolbars that were in previous versions of AutoCAD Civil are no longer available. The commands on those toolbars are now available from the ribbon.

**See also:**

- Transparent Commands (page 1581)

- *Quick Access Toolbar* in the AutoCAD User's Guide Help

**To display the AutoCAD Civil transparent command toolbars**

**1** Enter **-toolbar** at the command line.

**2** Enter **"transparent_commands"** or **"transparent_command_filters"** and then press Enter.

**3** Enter **Show**.

## Quick Reference

Command Line

-toolbar

# Working with AutoCAD Civil Windows

The Toolspace and the Panorama windows—the two main floating windows in AutoCAD Civil — share several features.
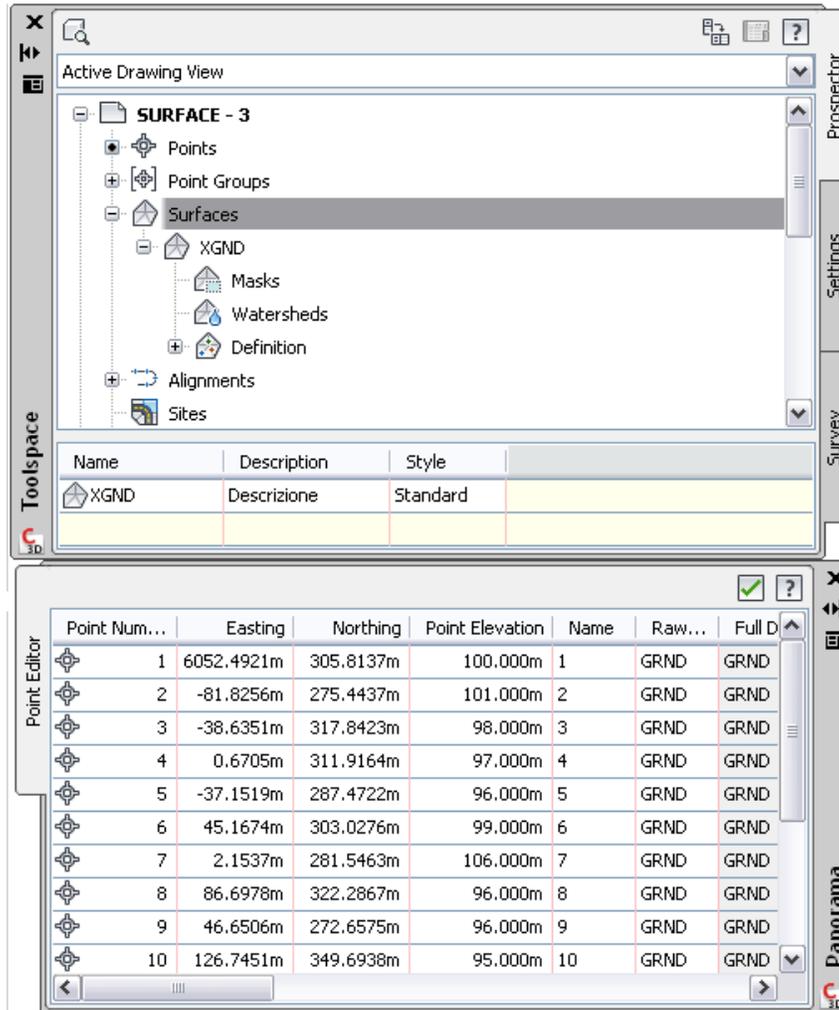


### Tabs

The Toolspace window always displays at least two tabs, Prospector and Settings. The Panorama window displays named tabs if you have more than one tool (vista) active in the window.

### Shortcut Menus

Right-click in the Toolspace or the Panorama window to display a shortcut menu of available commands. Right-click a single item, or select one or more items and right-click, to display a menu containing commands related to the item(s). If you right-click an area that contains no items or data, the menu contains commands related to the window.

### Auto-hide

This AutoCAD palette feature keeps the window active while maximizing the amount of available screen space. If Auto-hide is active for a window, the body of the window disappears when you move the cursor out of the window, leaving only the title bar visible. Move the cursor over the title bar to display the entire window again.

The following illustration shows the Toolspace window both closed and open, with Auto-hide active:
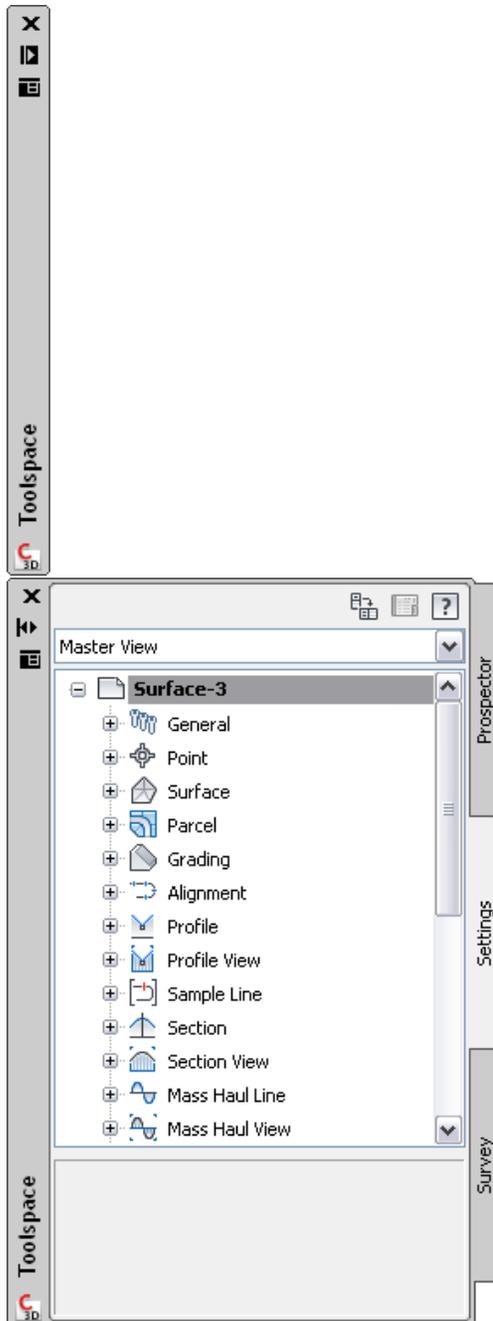


To activate Auto-hide for the Toolspace or the Panorama window, click  on the title bar. To deactivate Auto-hide, click  on the title bar. You can also right-click the title bar and use the shortcut menu to control Auto-hide. For more information about Auto-hide, see the AutoCAD Help.

**NOTE** Auto-hide is not available when a window is docked.

**Moving and Docking**

The Toolspace and the Panorama windows, like all AutoCAD palettes, can be moved and resized, and either floated in the window or docked. A docked window shares one or more edges with adjacent windows and toolbars. If a shared edge is moved, the windows change shape to compensate. To undock and relocate a window, click and drag the control bars at the top or side of the window. To prevent a window from docking while you drag it, hold down Ctrl.

**TIP** You can quickly undock a window by double-clicking the window's control bars.

For more information about working with docked and floating windows, see the AutoCAD Help.

# The Toolspace Window

The Toolspace window is an integral component in the user interface for accessing commands, styles, and data. Use it to access the Prospector, Settings, Survey, and Toolbox tabs. Right click each collection or item on these tabs to access commands.
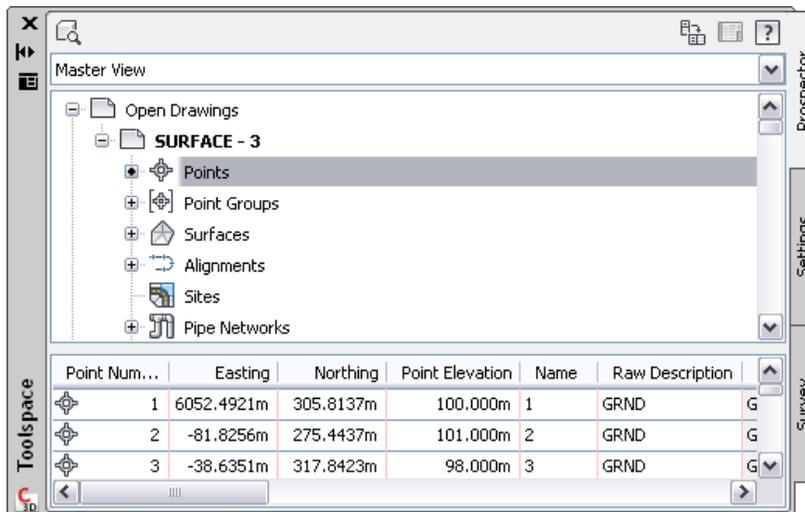


Use the Prospector tab to manage design objects. For more information, see The Toolspace Prospector Tab (page 87).

Use the Settings tab to manage object settings, styles, and other drawing items, such as point file formats, description key sets, and grading criteria sets. For more information, see The Toolspace Settings Tab (page 90).

Use the Survey tab to manage survey projects, data, and settings. For more information, see The Toolspace Survey Tab (page 97).

Use the Toolbox tab to access reports and subscription extensions, and to add custom tools. For more information, see The Toolspace Toolbox Tab (page 100).

# Displaying the Toolspace Window

You can control whether or not the Toolspace window is visible in the AutoCAD Civil session.

| If... | Then do this... |
|-------|-----------------|
| You cannot see the Toolspace window. | Click Home tab ➤ Palettes panel ➤ Toolspace.. |

| If... | Then do this... |
|---|---|
| Only the Toolspace title bar is displayed. | Move the cursor over the title bar to display the full Toolspace window. To disable Auto-hide, right-click the Toolspace title bar and click Auto-hide to remove the check mark next to it. |
| The contents of a tab are out of date. | Right-click in the window and click Refresh. |
| You want to close the Toolspace window. | Click ✕ . |

# Functions Shared by the Prospector, Settings, and Survey Tabs

The Prospector tab, Settings tab, and Survey tab have several controls in common, including a tree structure, shortcut menus, and object icons.
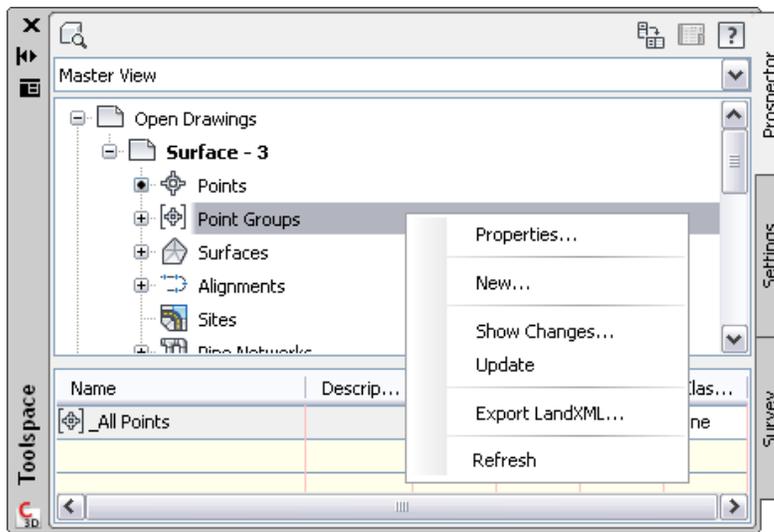


**Tree Structure**

The Toolspace trees display items in a hierarchical structure. Any item that can contain other items below it in the tree is a *collection*. For example, in the illustration, Open Drawings, Points-2, and Sites are all collections. The symbol displayed at the tree *node*, to the far left of each collection name gives you information about the display of the items in the collection:

| Symbol | Meaning and Possible Action |
|---|---|
| ⊞ | The collection contains items that are not displayed. Click ⊞ to expand the tree and display the items. |
| ⊟ | The items contained in the collection are displayed. Click ⊟ to collapse the tree and hide the items. |
| ◉ | The collection contains items that cannot be displayed in the tree. Click the collection name to list the items in a Toolspace item view. For more information, see The Toolspace Item View (page 82). |

| Symbol | Meaning and Possible Action |
|---|---|
| ⧄ | The collection's data needs to be refreshed. Right-click the collection and click Refresh. |
| ↗ | The item is a reference to an item located elsewhere in the tree. Double-click the reference to navigate to the actual item. For example, this icon appears next to a drawing in the Projects node when that project drawing is open. You can then double-click the icon to move to that drawing in the Open Drawings node in the Prospector tree. |
| ⋮ | The collection contains no items. |

**Shortcut Menus**

Display a shortcut menu for any collection or item in the tree by right-clicking the item. The menu contains commands that apply to the item you selected in the tree.

**Icons**

A row of icons appears at the top of each Toolspace tab. For more information about these, see The Prospector Tab Icons (page 87), The Settings Tab Icons (page 91) and the The Survey Tab Icons (page 99).

**Tree View**

Use the drop-down list above the tree to control which items are displayed in the tree. For more information, see Controlling the Items Displayed in the Settings Tree (page 91) and Controlling the Items Displayed in the Prospector Tree (page 88).

**Item Views**

When you select items in the tree, additional information may be displayed in an item view, which is shown in a pane next to or beneath the main tree. You can adjust the size of the item view by moving the bar that separates the item view from the tree.

An item view is one of the following, depending on the item selected:

- **List view.** Displays information in a list or table. In some cases, this information can be edited.

- **Graphical view.** Displays an image of the item.

For more information, see The Toolspace Item View (page 82).

**Copying Items in the Toolspace Trees**

You can copy some items in the Toolspace trees by dragging them to another drawing or collection. For information about using this feature in the Settings tree, see Dragging Items within the Settings Tree (page 97). You can use the Prospector tree drag-and-drop feature to add drawing points to drawing point groups. For more information, see Changing Point Group Properties by Dragging Points into the Point Group (page 536).

# The Toolspace Item View

Use the Toolspace item view to display information about an item or the contents of a collection.

The item view is a secondary pane displayed beside or below the tree. To display the item view, click the item name.

When the Toolspace window is floating, you can change the orientation of the item view by clicking the
🗗 button at the top of the Toolspace.

**NOTE** To see the item view, you may need to increase the size of the Toolspace window or move the bar that separates the item view from the Prospector, Settings, or Survey tree. To move the bar that separates the item view from the Prospector, Settings, or Survey tree, you must first select a collection that displays an item view.

An item view takes one of the following forms, depending on the selected item:

**List View**

A list view is a table displayed by the Toolspace trees for some items. In some cases, this information can be edited. The following illustration shows a Prospector list view of points:
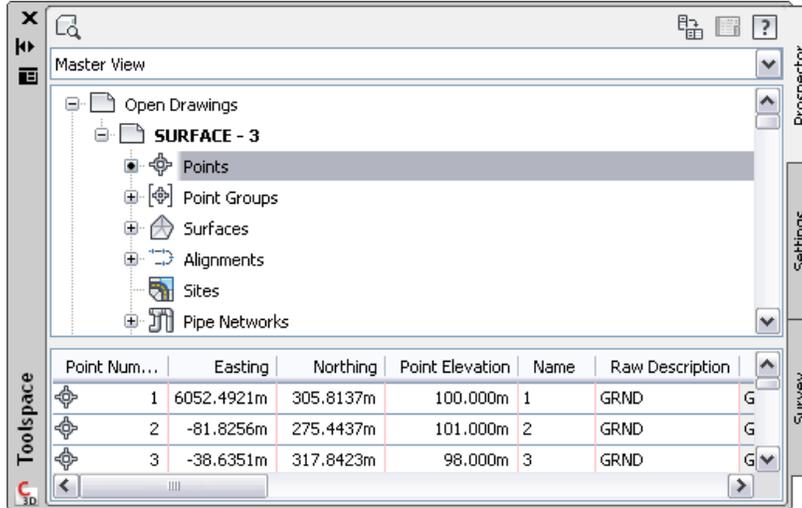


When the Toolspace is docked, the list view is displayed below the data tree. When the Toolspace is undocked, you can display the item view either below the data tree or to one side of the data tree by clicking the item view orientation button 🗗 .

**Graphical View**

A graphical view displays an image of the selected item. The following illustration shows a Prospector graphical view of a parcel.
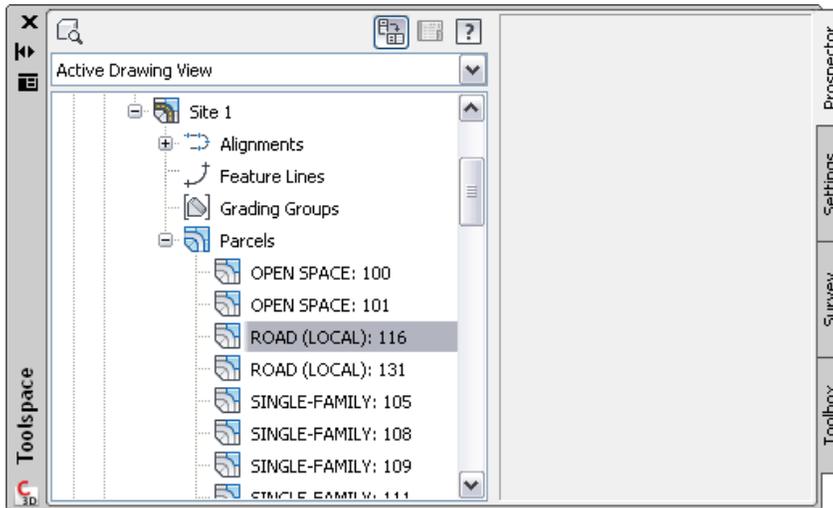
Right-click in the graphical preview to display a shortcut menu that allows you to manipulate the image using commands such as Pan and Zoom.

Not all items on the Prospector and Settings, or Survey trees have item views associated with them.

## Selecting Items in a List View

You can select one or more items in a Toolspace list view.

To work with an item in the list view, you must first select it in the list view by clicking the icon at the far left of the row. If you click within the row, you might select one cell but not the entire item.

**NOTE** You can quickly scroll to the top of the list view by selecting an item, holding down Ctrl and pressing Home. Scroll to the bottom of the list view by holding down Ctrl and pressing End.

**To select items in a list view**

1   In Toolspace, on the Prospector, Settings, or Survey tab, click an item that displays a list view.

    **NOTE** Some items do not have list views associated with them.

2   In the list view, click the icon at the left of the row that contains the first item you want to select.

3   Do one of the following:

    ■ To select additional contiguous items, hold down Shift and click the row containing the last item you want to select.

    ■ To select additional non-contiguous items, hold down Ctrl and click the items you want to select.

    ■ To select every item from the current item to the beginning of the list, hold down Shift and press Home.

    ■ To select every item from the current item to the end of the list, hold down Shift and press End.

## Customizing a List View

You can change the appearance of a list view by hiding columns, changing column positions, and changing column widths.

Right-click any column heading in the list view to display a shortcut menu that shows all the available columns. Checkmarks indicate which columns are currently displayed in the list view.

The value in a shaded column cannot be edited. If a displayed column is not in use, it contains a hatch pattern instead of data.

**To customize a list view**

**1**  In Toolspace, on the Prospector, Settings, or Survey tab, click an item that displays a list view.

> **NOTE** Some items do not have list views associated with them.

**2**  To show the names of all the columns that can be included in the list view, right-click any column heading.
Check marks indicate which columns are currently displayed in the list view.

**3**  To show all the columns in the list view, click Show All Columns.

**4**  To hide a column, do one of the following:
   ■  Right-click the column heading of the column you wish to hide and click Hide Column.

   ■  Right-click any column heading to display the column heading shortcut menu, and clear the check mark from the column you want to hide.

   The list view is displayed without the column.

**5**  To show a hidden column, right-click over any column heading to display the column heading shortcut menu, and select the column you want to show.

**6**  To change the position of a column in the list view, drag the column heading right or left.

**7**  To adjust the width of a column, move the cursor to the margin of the column heading until it changes to ✛ . Drag the column margin right or left.

## Changing the Contents of a Column in a List View

Use the column heading menu to change all the cells in a list view column to the same value.

For example, you can assign the same elevation to all the points in a list view.

**To change the contents of a column in a list view**

**1**  In Toolspace, on the Prospector, Settings, or Survey tab, click an item to display its list view.

> **NOTE** Some items do not have list views associated with them.

**2**  If any items in the list view are selected, press Esc to clear the selection.

**3**  Right-click the column heading of the column you want to change and click Edit.

**4**  In the edit box that is activated, enter the new value and press Enter.

## The List View Shortcut Menu

Right-click in a list view to display a shortcut menu.

Some commands apply only to items that you must select in the list view before right-clicking to display the menu. For more information, see Selecting Items in a List View (page 84).

The list view menu contains commands that apply to the selected items, plus the following standard commands if they are applicable:

| Select this menu item... | If you want to do this.... |
| --- | --- |
| Zoom To | (Drawing items only) Zoom the drawing to the selected item or items in the list view. |
| Pan To | (Drawing items only) Pan the drawing to the selected item or items in the list view. |
| Line Shading | Display the list view with a shading guide every three rows. |
| Copy To Clipboard | Copy a tab-delimited copy of the selected information to the clipboard so that you can paste it into a document for printing or editing. For more information, see Copying Items from a List View (page 86). |
| Copy Value To Clip-board | Copy the contents of the specific value in the cell. This command copies the contents of a selected cell as a raw text (floating point) value. For example, if you use this command to copy station data displayed as 2+00.00 in the list view, the value is copied to the clipboard as 200.00. |

## Copying Items from a List View

Use the Copy To Clipboard or Copy Value To Clipboard commands on the list view shortcut menu to copy the contents of an item view or single cell value to a file for editing or printing.

The Copy To Clipboard command copies a tab-delimited list of all the selected items in the list view, including the column headers, to the Windows clipboard. You can then paste the information into an Excel spreadsheet or a text file.

The Copy To Clipboard command copies all displayed columns, including columns without values. It does not copy hidden columns.

The Copy Value To Clipboard command copies the contents of the cell that your mouse is currently hovering over as a raw text (floating point) value.

For example, if you use this command to copy station data displayed as 2+00.00 in the list view, the value is copied to the clipboard as 200.00.

Each value is copied with the same precision used to display the value in the list view. You can change the precision of some values using the drawing and feature level settings. For more information, see Understanding Settings (page 61).

**To copy items from a list view**

1 Select the items you want to copy. For more information, see Selecting Items in a List View (page 84).

2 Right-click to display the list view shortcut menu for the selected items.

3 Click Copy To Clipboard.

4 Open a prepared Excel spreadsheet or a text file and paste the clipboard contents into the file.

# The Toolspace Prospector Tab

You can use the Prospector tab to manage project and drawing objects.
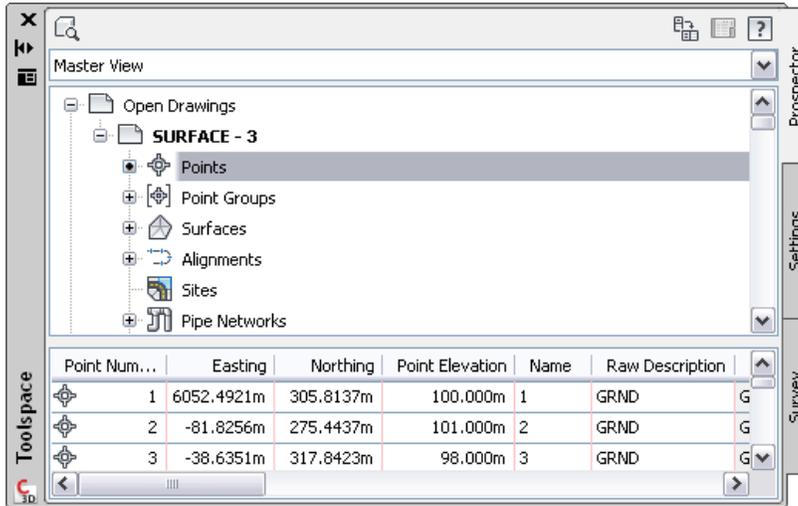


The Prospector tab is a default tab in the Toolspace window and is available if the Toolspace is displayed (page 80).

On this tab, all of the objects in a drawing or project are arranged in a hierarchy that you navigate in standard, Windows Explorer–like fashion.

**To access the Prospector tab**

■ Click Home tab ➤ Palettes panel ➤ Toolspace ⚒ .

The Prospector tab and the Settings tab share some basic functions. For more information, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

The Prospector tree contains the following top-level collections, which can be viewed using the Master View:

■ **Open Drawings.** For more information about this collection, see The Prospector Tree (page 88).

■ **Projects.** For more information about this collection, see Project Management (page 125).

> **NOTE** Projects lists projects available after you have logged into an Autodesk Vault server and Vault database.

■ **Drawing Templates.** For more information about this collection, see Drawing Templates (page 109).

## The Prospector Tab Icons

Use the icons at the top of the Prospector tab to control the display of icons within the Prospector tree, to display the Panorama window, and to access Help.

| Icon | Purpose |
| --- | --- |
| ⌕ | Toggles the Toolspace item preview display. Click to enable previews. When item previews are disabled, the Show Preview menu item in an item's shortcut menu is disabled. |
| ⬚ | Switches the orientation of the item and list views. Available only when the Toolspace is undocked. |

| Icon | Purpose |
|---|---|
|  | Switches the display of the Panorama window. If the Panorama window doesn't contain any active vistas, this button is not available. For more information, see The Panorama Window (page 102). |
|  | Displays Help. |

## Controlling the Items Displayed in the Prospector Tree

Use the drop-down list at the top of the Prospector tab to control which items are displayed in the Prospector tree.



- **Master View.** Displays all project and drawing items, including drawing templates. The name of the active drawing is highlighted.

- **Active Drawing View.** Displays only items in the active drawing. If you switch to another drawing, the tree is updated to reflect the new drawing.

## The Prospector Tree

Use the Prospector tree to manage and access drawing and project objects.

The Prospector tree and the Settings tree share some basic functions. For more information about basic Toolspace tree operations, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

The following illustration shows an expanded Prospector tree for a drawing:
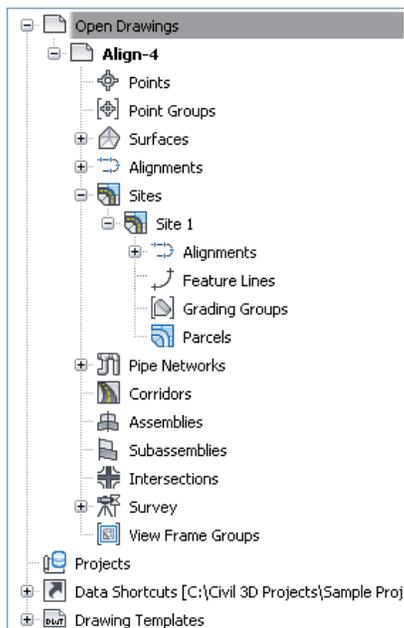
Refer to the following table for information about the drawing collections in the Prospector tree:

| For information about this item in the tree... | See.... |
| --- | --- |
| Points | Points Collection (Prospector Tab) (page 409) |
| Points Groups | Point Groups Collection (Prospector Tab) (page 522) |
| Surfaces | Surfaces Collection (Prospector Tab) (page 600) |
| Alignments | Alignment Collection (Prospector Tab) (page 856) |
| Sites | Sites Collection (page 720) |
| Pipe Networks | Pipe Networks Collection (Prospector Tab) (page 1188) |
| Corridors | Corridors Collection (Prospector Tab) (page 1330) |
| Assemblies | Assemblies Collection (Prospector Tab) (page 1379) |
| Subassemblies | Subassemblies Collection (Prospector Tab) (page 1395) |
| Intersections | Intersection Collection (Prospector Tab) (page 1426) |
| Survey | Survey Collection (Prospector Tab) (page 193) |
| View Frame Groups | View Frame Groups Collection (Prospector Tab) (page 1694) |
| Feature Lines | Feature Lines Collection (Prospector Tab) (page 733) |
| Parcels | Assemblies Collection (Prospector Tab) (page 1379) |
| Profiles | Profiles Collection (Prospector Tab) (page 1022) |
| Profile Views | Profile Views Collection (Prospector Tab) (page 1022) |
| Sample Lines | Sample Line Groups Collection (Prospector Tab) (page 1125) |
| Sections | Sections Collection (Prospector Tab) (page 1126) |
| Section Views | Section View Groups Collection (Prospector Tab) (page 1126) |
| Grading Groups | Grading Groups Collection (Prospector Tab) (page 734) |
| Data Shortcuts | Using Data Shortcuts (page 127) |

Standard shortcut menu commands are available for many Prospector tree items and collections. To display the shortcut menu, right-click a collection or item in the Prospector tree. The following commands are available on the Prospector shortcut menu whenever they are applicable to the selected item(s):

| Select this shortcut menu command... | If you want to do this.... |
| --- | --- |
| Properties | View or edit the properties of the selected item or items. |
| Edit | Edit the selected item or items. |

| Select this shortcut menu command... | If you want to do this.... |
|---|---|
| Delete | Delete the selected item or items. |
| Select | Selects the item or items in the drawing. |
| Zoom To | Zoom the drawing to the selected item or items contained in the collection. |
| Pan To | Pan the drawing to the selected item or items contained in the selected collection |
| Export LandXML | Export information using LandXML. |
| Refresh | Update the items displayed in the Prospector tree. |

**NOTE** Some Prospector shortcut menus do not include all the commands in the preceding table. Also, a Prospector shortcut menu can contain additional commands that are related to the selected collection or item, including project management access control commands.

The Prospector tree contains collections for projects and drawing templates. For more information, see Project and Drawing Items in the Prospector Tree (page 139).

## The Toolspace Settings Tab

You can use the Settings tab to manage styles for AutoCAD Civil objects and to control settings for drawings and commands.
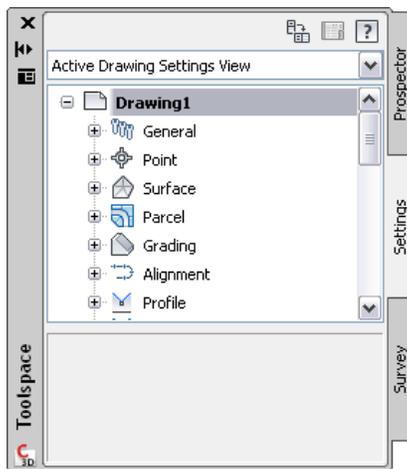


The Settings tab is a default tab in the Toolspace window and is available if the Toolspace is displayed (page 80).

On this tab, styles are organized for different object types. Even in a blank drawing, most of these styles are present in a standard hierarchy. You can create and modify styles in a drawing, then save it as a template. Subsequent drawings based on the template will automatically have the same set of styles available. You can modify object, label, and table styles. You can also control settings for drawings and commands.

**To access the Settings tab**

■ Click Home tab ➤ Palettes panel ➤ Toolspace ⚒ .

The Prospector tab and the Settings tab share some basic functions. For more information about basic Toolspace tree operations, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

## The Settings Tab Icons

Use the icons at the top of the Settings tab to control the display of icons within the Settings tree, to display the Panorama window, and to access Help.

| Icon | Purpose |
| --- | --- |
| | Toggles the orientation of the item view when the Toolspace window is floating. For more information, see The Toolspace Item View (page 82). |
| | Switches the display of the Panorama window. If the Panorama window doesn't contain any vistas, this control is not active. For more information, see The Panorama Window (page 102). |
| | Displays Help. |

## Controlling the Items Displayed in the Settings Tree

Use the drop-down list at the top of the Settings tab to control which items are displayed in the Settings tree.

| Active Drawing Settings View |
| --- |
| Master View |
| Active Drawing Settings View |
| Active Drawing Labels Only View |
| Labels Only View |
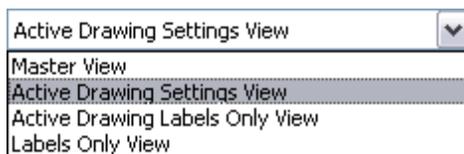
Select one of the following tree views on the Settings tab:

■ **Master View.** Displays items for all open drawings. The name of the active drawing is shown in boldface.

■ **Active Drawing Settings View.** Displays only the items for the active drawing.

■ **Active Drawing Labels Only View.** Displays only label style collections and label styles items for the active drawing.

■ **Labels Only View.** Displays only label style collections and label style items for all drawings.

## The Settings Tree

Use the Settings tree to manage object, label, and table styles and to control settings for drawings and commands.

For more information about basic Toolspace tree operations, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

The following illustration shows the top-level collections in the Settings tree:
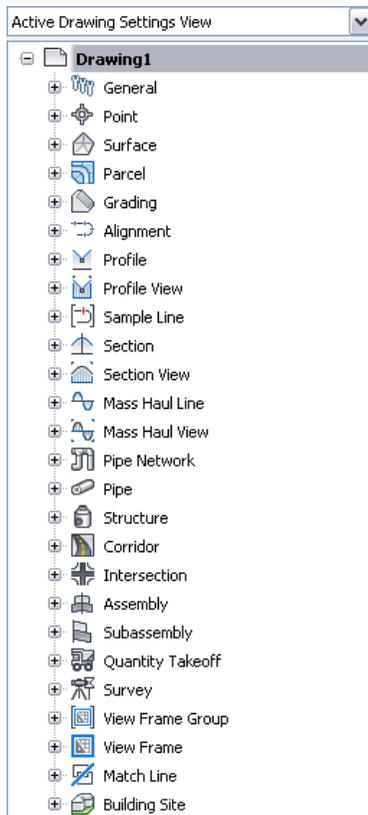
| For information about this item in the tree... | See... |
|---|---|
| General | General Collection (Settings Tree) (page 94) |
| Point | Point Collection (Settings Tab) (page 409) |
| Surface | Surface Collection (Settings Tab) (page 601) |
| Parcel | Parcel Collection (Settings Tab) (page 812) |
| Grading | Grading Collection (Settings Tab) (page 735) |
| Alignment | Alignment Collection (Settings Tab) (page 858) |
| Profile | Profile Collection (Settings Tab) (page 1022) |
| Profile View | Profile View Collection (Settings Tab) (page 1023) |
| Sample Line | Sample Line Collection (Settings Tab) (page 1127) |
| Section | Section Collection (Settings Tab) (page 1127) |
| Section View | Section View Collection (Settings Tab) (page 1128) |
| Mass Haul Line | Editing Mass Haul Line Settings (page 1178) |
| Mass Haul View | Editing Mass Haul View Settings (page 1179) |

| For information about this item in the tree... | See... |
|---|---|
| Pipe Network | Pipe Network Collection (Settings Tab) (page 1188) |
| Pipe | Pipe Collection (Settings Tab) (page 1189) |
| Structure | Structure Collection (Settings Tab) (page 1189) |
| Corridor | Corridor Collection (Settings Tab) (page 1330) |
| Intersection | Intersection Collection (Settings Tab) (page 1428) |
| Assembly | Assembly Collection (Settings Tab) (page 1380) |
| Subassembly | Subassembly Collection (Settings Tab) (page 1396) |
| Quantity Takeoff | Quantity Takeoff Settings (page 1161) |
| Survey | Survey Collection (Settings Tab) (page 194) |
| View Frame Group | View Frame Group Collection (Settings Tab) (page 1694) |
| View Frame | View Frame Collection (Settings Tab) (page 1696) |
| Match Line | Match Line Collection (Settings Tab) (page 1698) |
| Building Site | Edit Feature Settings - Import Building Site (page 1793) |

Most of the object collections above conform to a standard layout and use a standard shortcut menu.

## The Object Collection (Settings Tree)

Use the object collection in the Settings tree to access object styles, settings and drawing-related information for a class of objects.

The following illustration shows an example of a Settings tree object collection:
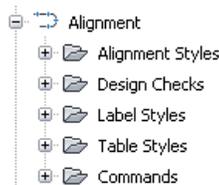


Right-click the object collection to display a shortcut menu with the following commands:

| Select this command... | If you want to... |
|---|---|
| Edit Feature Settings | Modify the settings for all commands that pertain to the object. |
| Edit Label Style Defaults | Set default settings for all label styles belonging to the object. |
| Refresh | Update the Settings tree display. |

For detailed information about the settings, click Help in the dialog box that is displayed when you select the menu item.

## The Object Style Collection (Settings Tree)

Use an object's Styles collection in the Settings tree to manage styles for a class of objects.

The following illustration shows an example of a Settings tree object style collection:



Right-click the style collection to display a standard shortcut menu with the following commands:

| Select this command... | If you want to... |
| --- | --- |
| New | Create a new style. |
| Refresh | Update the Settings tree display. |

Right-click a style name to display a standard shortcut menu with the following commands:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Change the properties of the selected style. |
| Copy | Create a new style based on an existing style. |
| Delete | Delete the selected style. This command is displayed only when the style can be deleted. |
| Refresh | Update the Settings tree display. |

## General Collection (Settings Tree)

Use the General collection to create multipurpose and label styles that can be used by more than one object type.

Pipes and Surface objects can share the Render styles in the General Multipurpose Styles collection, while Corridors and Grading share the other styles in this folder.

The General Label Styles are used by lines, curves, feature lines, and corridors. This collection also contains Note label styles, which are not specific to an object.

The following illustration shows an illustration of the General collection:
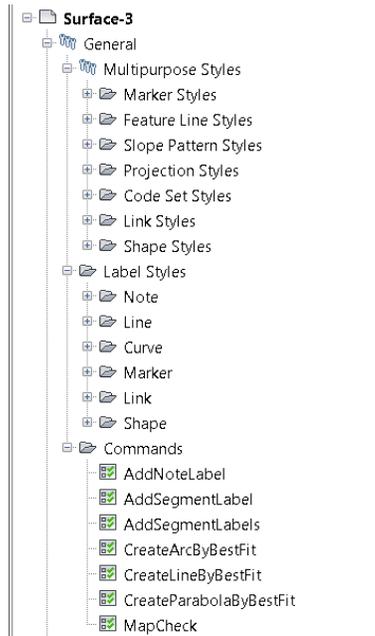


Right-click a style name to display a shortcut menu that contains the following commands:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Change the properties of the style. |
| Copy | Create a new style based on an existing style. |
| Delete | Delete the selected style. This command is displayed only when the style can be deleted. |
| Refresh | Update the Settings tree display. |

For detailed information about the settings, click Help in the dialog box that is displayed when you select the menu item.

## The Label Styles Collections (Settings Tree)

Use the Label Styles collections in the Settings tree to create and manage label styles and expressions for a class of objects.

Click the +/- box to expand the collection and see the label styles for the object. Some objects have multiple levels of label styles.

The following illustration shows an example of a label style collection:
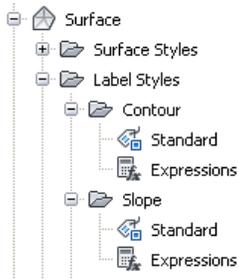
Right-click a label style name to display a shortcut menu that contains the following commands:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Change the properties of the label style. |
| Copy | Create a new label style based on an existing label style. |
| Delete | Delete the selected label style. This command is displayed only when the style can be deleted. |
| Refresh | Update the Settings tree display. |

For detailed information about the settings, click Help in the dialog box that is displayed when you select the menu item.

## The Table Styles Collections (Settings Tree)

Use the Table Styles collections in the Settings tree to create and manage table styles for a class of objects.

Click the +/- box to expand the collection and see the table styles for the object. Some objects have multiple levels of table styles.

The following illustration shows an example of a Settings tree table style collection:
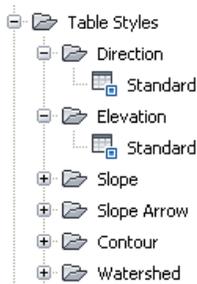


Right-click a table styles collection to display a shortcut menu with the following commands:

| Select this command... | If you want to... |
| --- | --- |
| New | Create a new table style. |
| Refresh | Update the Settings tree display. |

For detailed information about the settings, click Help in the dialog box that is displayed when you select the menu item.

Right-click a table style name to display a shortcut menu that contains the following commands:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Change the properties of the table style. |
| New | Create a new table style. |
| Copy | Create a new table style based on an existing table style. |
| Delete | Delete the table style. This command is displayed only when the style can be deleted. |
| Refresh | Update the Settings tree display. |

For detailed information about the settings, click Help in the dialog box that is displayed when you select the menu item.

## The Commands Collections (Settings Tree)

Use the Commands collections in the Settings tree to control the settings for a specific command at the object (feature) level.

When you change settings using the Commands collections, the changes affect a single command only. For more information, see Understanding Settings (page 61).

The following illustration shows an example of a Settings tree commands collection:
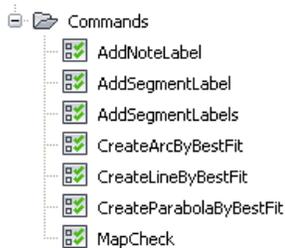


Right-click a command name and click Edit Command Settings to display the Edit Command Settings dialog box for the command. You can then change the settings.

## Dragging Items within the Settings Tree

Copy styles, description keys, and pipe rules using the Settings tree drag-and-drop capability.

For more information on dragging items within or between drawings, see Dragging Items Within or Between Drawings (page 119).

## The Toolspace Survey Tab

You can use the Survey tab to manage survey user and system settings as well as survey data.
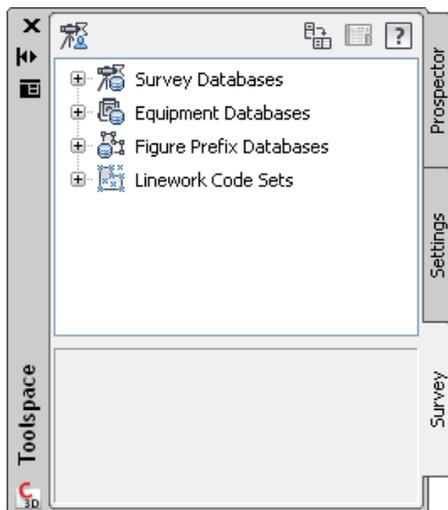
This tab displays survey project data, organized within databases for survey projects, equipment, and figure prefixes. The project databases record the survey points, networks, and figures. The equipment databases record standard deviations and other operational parameters of individual pieces of survey equipment. The figure prefix databases record the conversion routines that are applied when creating lots, buildings, or other figures from survey points.

The contents of the Survey tab are not specific to a drawing. This tab reflects the survey data in your AutoCAD Civil Projects folder, so it facilitates access to survey data from multiple drawings.

The surveyed points and figures in a project can be converted to AutoCAD Civil points and parcels.

The Survey tab and Prospector tab share some basic functions. For more information, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

The Survey tab can be toggled on/off from view in the AutoCAD Civil Toolspace by clicking  Home tab ➤

Palettes panel ➤ 𝐀𝐓 Survey Toolspace.

The Survey tree provides access to survey data and settings.

## Displaying and Hiding the Survey Toolspace Tab

Use the Survey tab in the Toolspace to access and manage survey feature system settings and survey project data.

Survey data is managed on the Survey tab of Toolspace instead of on the Prospector tab like other objects.

This is a persistent setting and if you click Close, the Survey tab will not appear on the Toolspace the next time you start AutoCAD Civil.

**To display the Survey tab**

■   Click Home tab ➤ Palettes panel ➤ Survey Toolspace 𝐀𝐓

**To hide the Survey tab**

■   Click Home tab ➤ Palettes panel ➤ Survey Toolspace 𝐀𝐓

**Quick Reference**

Ribbon

Home tab ➤ Palettes panel ➤ Survey Toolspace ⛏

Command Line

OST or CST

## The Survey Tab Icons

Use the icons at the top of the Survey tab to control the display of icons within the Survey tree, to display the Panorama window, and to access Help.

| Icon | Purpose |
|---|---|
| ⛏ | Displays the Survey User Settings dialog box. User settings are specific to a Windows user login account and affect the Survey feature system, not database or drawing data. |
| 🗃 | Switches the orientation of the item and list views. Available only when the Toolspace is undocked. |
| 🗐 | Switches the display of the Panorama window. If the Panorama window doesn't contain any active vistas, this control is not available. For more information, see The Panorama Window (page 102). |
| ? | Displays Help. |

## The Survey Tree

Use the Survey tree to manage and access survey settings and data.

The Survey tree and the Prospector tree share some basic functions. For more information about basic Survey tree operations, see Functions Shared by the Prospector, Settings, and Survey Tabs (page 81).

The Survey tree provides access to survey data and settings:
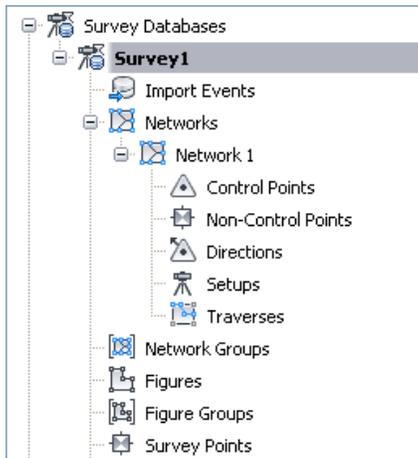
- **Survey Databases.** A survey database collection expands to display all the networks (page 201) (which collect the control points (page 253), non-control points (page 256), known directions (page 283), observations (page 263), setups (page 259), and traverse definitions (page 373)), figures (page 321), and survey points.

- **Equipment Databases.** Expands to display available equipment database. For more information about this collection, see Survey Equipment Database (page 199).

- **Figure Prefix Databases.** Expands to display available figure prefix databases. Figure prefix databases enable you to determine the layer that a figure is drawn on by creating a prefix for a group of figure names. For more information about this collection, see Survey Figure Prefix Database (page 200).

- **Linework Code Sets.** Expands to display available Linework Code Sets. Linework code sets are used to interpret the syntax of the field codes that are entered into the data collector by the survey field crew. For more information about this collection, see Linework Code Sets (page 230).

Standard shortcut menu commands are available for many Survey tree items and collections. To display the shortcut menu, right-click a collection or item in the Survey tree.

## The Toolspace Toolbox Tab

You can use the Toolbox tab to access the Reports Manager and subscription extensions and to add custom tools.

The Toolbox tab of Toolspace is the primary interface for running reports and extensions. Like the Survey tab, you can show or hide the Toolbox tab as needed.

### Reports

The Toolbox tab organizes reports for each object type. The reports provide useful engineering data from a drawing in a compact, portable format. Some are in LandXML format, with predefined or custom XSL style sheets. Other reports are .NET format, with custom dialog boxes that allow you to select the data and various options.

You can add your own reports to the Toolbox menu. These reports can be in a variety of formats, including XML, VBA, COM, or .NET.

### Subscription Extensions

When you install subscription extensions, they are added to the Subscription Extension Manager collection in the Toolbox.
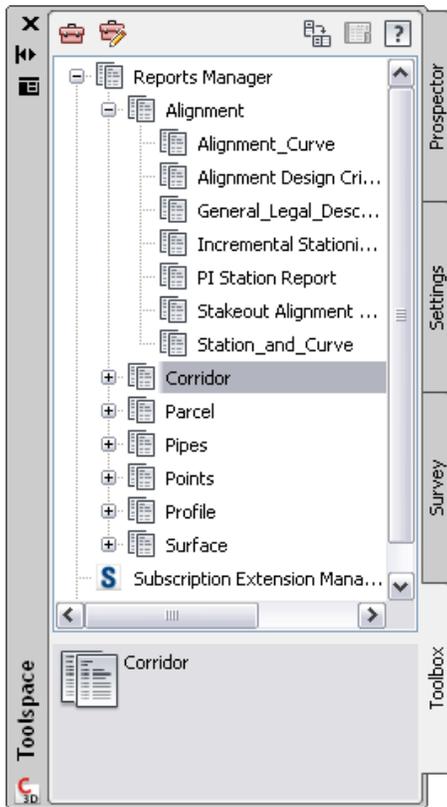
**See also:**

■ Reports (page 1573)

## Displaying and Hiding the Toolbox Tab

The Toolbox can be turned on and off using the same menu command.

**To display and hide the Toolbox tab**

■ Click Home tab ➤ Palettes panel ➤ ▼ ➤ Toolbox 💼 .

## Quick Reference

Ribbon

  Home tab ➤ Palettes panel ➤ ▼ ➤ Toolbox 💼

Menu

  General menu ➤ Toolbox

Command Line

  ToolBox

## The Toolbox Tab Icons

Use the icons at the top of the Toolbox tab to edit report settings, to display the Toolbox Editor, to display the Panorama window, and to access Help.

| Icon | Purpose |
| --- | --- |
| | Opens the Edit Report Settings (page 2212) dialog box, where you can edit settings for all report types. |
| | Opens the Toolbox Editor (page 2211), where you can add custom reports. You can also add other custom tools. |
| | Switches the display of the Panorama window. If the Panorama window doesn't contain any active vistas, this control is not available. For more information, see The Panorama Window (page 102). |
| | Displays Help. |
| | Displays available report types. Expand the Reports Manager tree and double-click a report type to run it. Both LandXML and VBA reports are listed in this tree and are distinguished by their icons. |

## The Toolbox Tree

Use the Toolbox tree to access the reports available in the Reports Manager, to access Subscription Extensions, and to access other custom tools.

The following commands are available on the Toolbox shortcut menu whenever they are applicable:

| Select this shortcut menu command... | If you want to do this.... |
| --- | --- |
| Execute | Run a selected report. |
| Refresh | Update the items displayed in the Toolbox tree. |

# The Panorama Window

Use the Panorama window to display horizontally oriented data, such as point data, in a grid.

The Panorama window can display many types of data, such as the Point Editor vista and the Alignment Entities vista. To conserve screen space if more than one vista is active, the Panorama window displays a tab for each one, as shown in the following illustration.

| Point Num... | Easting | Northing | Point Elevation | Name | Raw Description | Full Description |
|---|---|---|---|---|---|---|
| 1 | 19857.2577' | 20260.4556' | 100.306' | | GRND | GRND |
| 2 | 19875.4801' | 20207.7198' | 101.093' | | GRND | GRND |
| 3 | 19887.3953' | 20692.7130' | 96.419' | | GRND | GRND |
| 4 | 19888.9069' | 21076.9713' | 95.436' | | GRND | GRND |
| 5 | 19889.7110' | 21205.2833' | 97.833' | | GRND | GRND |
| 6 | 19890.2416' | 21136.5660' | 96.612' | | GRND | GRND |
| 7 | 19891.2471' | 20986.9661' | 97.474' | | GRND | GRND |
| 8 | 19892.5010' | 20942.4705' | 97.856' | | GRND | GRND |
| 9 | 19893.3261' | 21033.2971' | 95.738' | | GRND | GRND |
| 10 | 19895.8462' | 20756.7242' | 98.217' | | GRND | GRND |

Click the tab containing a vista's name to bring it to the front of the window.

You control the display of some vistas, such as the Point Editor vista. You activate them when you want, and they remain active until you dismiss them by clicking . Other vistas, such as the Alignment Entities vista, are controlled by a specific command. This type of vista can only be activated or closed when the command is running, and its display is controlled from the command toolbar.

The active drawing controls the state of the Panorama window. When you switch between drawings, the Panorama window either becomes inactive or shows only the vistas associated with the active drawing.

You can dock the Panorama window or use Auto-hide to reduce the amount of screen space it requires. For more information, see The Toolspace Window (page 80).

## Displaying the Panorama Window

Display of the Panorama window is affected by several factors, such as the active drawing and the active vistas within that drawing.

If the Panorama window is not visible, refer to the following table for possible reasons and actions you can take to display it:

| Under these circum-stances... | Do this... |
|---|---|
| You cannot see the Panorama window or the title bar. | Click at the top of the Toolspace window. If the Panorama window contains active vistas, it is displayed. If the button is disabled, the Panorama window contains no active vistas. In some cases, you can display the Panorama window by clicking a toolbar icon. |
| The Panorama window is displayed, but the vista you want to work with is not displayed. | If the Panorama window has a tab for the vista, click the tab to display the vista. If the named tab is not visible, you must activate the vista. For instructions about activating a specific vista, refer to Help for that vista. |
| Only the Panorama title bar is displayed (Auto-hide is active). | Move the cursor over the title bar to display the full Panorama window. To disable Auto-hide, click on the Panorama title bar. |

# Layout Toolbars

Layout toolbars let you create and or edit AutoCAD Civil objects.

AutoCAD Civil has the following layout toolbars:

■ Alignment Layout Toolbar

■ Grading Creation Toolbar

■ Parcel Layout Toolbar

■ Pipe Network Layout Toolbar

■ Point Object Creation Toolbar

■ Profile Layout Toolbar

The icons or commands available on an AutoCAD Civil layout toolbar depend upon the toolbar in use.

Most AutoCAD Civil layout toolbars include the following three icons in the upper right corner:

| Icon | Purpose |
| --- | --- |
|  | Pins the toolbar, which places it in a fixed location and also shrinks the toolbar down to a small placeholder when the cursor moves outside the toolbar. To display the expanded toolbar, move the cursor over the placeholder. To unpin the toolbar, click  . |
|  | Displays Help for the layout toolbar. |
|  | Closes the layout toolbar. |

For information about the icons and commands that are available from a specific layout toolbar, refer to the following table, or click the Help button on the toolbar.

| For information on the... | see |
| --- | --- |
| Alignment Layout Toolbar | Alignment Layout Tools (page 1747) |
| Grading Object Creation Toolbar | Grading Creation Tools (page 1892) |
| Parcel Layout Toolbar | Parcel Layout Tools (page 2009) |
| Pipe Network Layout Toolbar | Network Layout Tools (page 2031) |
| Point Object Creation Toolbar | Create Points Dialog Box (page 2117) |
| Profile Layout Toolbar | Profile Layout Tools (page 2170) |

# Tool Palettes

Tool palettes provide convenient access to a variety of tools and content.

Tool palettes are a standard feature in AutoCAD. You can open tool palettes using the ToolPalettes command, or from the ribbon.

Tool palettes provide access to both AutoCAD and AutoCAD Civil tools and content. For example, the AutoCAD Civil tool palettes provide access to subassemblies and assemblies that you can use in corridor design. They also provide access to civil engineering related render materials and multi-view blocks.

The AutoCAD tool palettes provide access to items such as AutoCAD render materials, AutoCAD multi-view blocks, and AutoCAD visual styles.

You can drag and drop an item from a Tool Palette into the drawing to insert it. Any prompts that are required for inserting the item are displayed at the command line.

Tool palettes can be customized to suit your needs. For more information on working with tool palettes, see Tool Palettesin AutoCAD Help.

For information about adding render materials to a drawing from a tool palette, see Adding Materials to Drawings (page 1578).

For more information about working with the Content Browser and tool palettes, see Adding Content to a Tool Catalog in the Content Browser Help.

**To display the Tool Palettes Window and select a palette**

1   Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

2   In the Tool Palettes Window, on the title bar, click the Properties icon  .

3   From the bottom of the menu, select the tool palette to display.

**To add a subassembly or assembly to a tool palette**

1   Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

2   Switch to the tool palette you want to add the subassembly or assembly to, or create a new tool palette.

3   Click View tab ➤ Palettes panel ➤ Content Browser  .

4   In the Content Browser, locate the subassembly or assembly you want to add to a tool palette.

5   Click the i-drop icon on the subassembly or assembly.



6   Hold down your pointing device and drag the subassembly or assembly to the location in the tool palette where you want to place it.



Now you can use the procedure described in Creating a Subassembly from a Tool Palette (page 1396) to add this subassembly to a drawing. For more information about working with the Content Browser and tool palettes, see Adding Content to a Tool Catalog in the Content Browser Help.

## Quick Reference

Ribbon

Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

Menu

   Click General menu ➤ Tool Palettes Window.

Command Line

   ToolPalettes

# Content Browser

The Content Browser is a library of tool catalogs containing tools, tool palettes, and tool packages. You can publish catalogs so that multiple users have access to standard tools for projects.

Metric and imperial subassembly tool catalogs are included. You can drag and drop subassemblies from a catalog onto a tool palette for easier access.

For more information, see Understanding the Content Browser in the Content Browser Help.

**To display the Content Browser**

■   Click View tab ➤ Palettes panel ➤ Content Browser 🔳 .

### Quick Reference

Ribbon

   Click View tab ➤ Palettes panel ➤ Content Browser 🔳 .

Menu

   Click General menu ➤ Content Browser.

Command Line

   ContentBrowser

# Object Tooltips

Proximity and rollover tooltips are displayed for civil drawing objects. You can control the display of tooltips for the drawing, for an object type, or for a specific object.

■   A rollover tooltip displays information about an object when you hover the cursor directly over it. If the object highlighting setting is on, the object will be highlighted when this tooltip is displayed.

■   A proximity tooltip displays information about the current location of the cursor with respect to one or more objects in the drawing. More specifically, it displays information about the proximity of the cursor to the objects. These tooltips are displayed without having to hover the cursor over any specific entity in the drawing.

The most important object information is displayed at the beginning of the tooltip. For alignments and surfaces, the tooltip displays the station/offset information for the closest alignment to the cursor and the elevation of the topmost surface.

If you keep your cursor in the same location, additional information is displayed in an expanded tooltip that shows up to four more items. Which items are displayed depends upon the objects in the drawing and whether tooltips are enabled for that object (page 107). If there are five alignments and five surfaces in the drawing, for example, the proximity tooltip will display three alignments and three surfaces.

Alignments are shown in order of proximity to the cursor. Surfaces are shown in order from highest elevation first, to lowest elevation last.

**To control the display of tooltips for the drawing**

■ Click Application Menu ➤ Options. On the Display tab, select or clear the Show ToolTips and Show Rollover ToolTips check boxes.

> **NOTE** The Show ToolTips check box controls the display of tooltips for the ribbon and for dialog boxes, as well as proximity tooltips. The Show Rollover ToolTips check box controls the display of AutoCAD and AutoCAD Civil rollover tooltips.

**To control the display of tooltips for an object type**

1 On the Toolspace Settings tab, right-click <object> ➤ Edit Feature Settings.

2 In the Edit Feature Settings dialog box, under General, edit the Show Tooltips value.

> **NOTE** This setting can be overridden at the object level in the object properties dialog boxes.

  ■ **Yes**: Displays tooltips for all objects of the selected type by default.

  ■ **No**: Does not display tooltips for the selected object type.

3 Edit the New Entity Tooltip State.
  ■ **On**: Sets the default state of tooltips for new objects to be turned on.

  ■ **Off**: Sets the default state of tooltips for new objects to be turned off.

4 Click OK.

**To control the display of tooltips for an object**

■ In any civil object properties dialog box, on the Information tab, select or clear the Show Tooltips check box.

# User Interface Command Reference

This section provides brief descriptions of the user interface-related commands available from the command line in AutoCAD Civil.

Use the links in the following table to access more information about user interface commands available from the command line in AutoCAD Civil:

| Command | Description |
| --- | --- |
| ContentBrowser | Displays the Content Browser. |
| CST | Closes the Toolspace Survey tab (page 98). |
| DockPanorama | Docks the Panorama window (page 102). |
| DockTS | Docks the Toolspace window (page 80). |
| FloatPanorama | Changes the Panorama window from docked to floating (page 102). |

| Command | Description |
| --- | --- |
| FloatTS | Changes the Toolspace window from docked to floating (page 80). |
| HidePanorama | Hides the Panorama window (page 103). |
| HideTS | Hides the Toolspace window (page 80). |
| Menubar | Displays the legacy menu bar. (page 76) |
| OST | Opens the Toolspace Survey tab (page 98). |
| Ribbon | Displays the ribbon (page 71) |
| RibbonClose | Hides the ribbon. (page 71) |
| ShowPanorama | Displays the Panorama window (page 103). |
| ShowTS | Displays the Toolspace window (page 80). |
| ToolBox | Displays the Toolbox window (page 101). |
| ToolPalettes | Displays the Tool Palettes Window (page 104). |

# Working with Drawings

<div style="text-align: right; font-size: 3em; font-weight: bold;">6</div>

AutoCAD Civil objects are stored in drawings by default.

If you work in a multi-user environment or want to manage your drawings by having different objects in different drawings you will want to use Project Management. For more information, see Project Management (page 125).

**NOTE** AutoCAD Civil uses the standard AutoCAD drawing functionality. For more information, see the help topics.

Use the list at the top of the Prospector tab to control whether only the active drawing is listed in the prospector tree or all open drawings are listed.

## Drawing Templates

Use drawing templates to eliminate duplication of effort and to help maintain consistency across drawings.

You start a new AutoCAD Civil drawing using a template file. An AutoCAD Civil drawing template can contain standard AutoCAD information, such as AutoCAD settings and layers, and AutoCAD objects, such as lines and text. In addition, it can include any AutoCAD Civil drawing information that is listed in either the Settings tree (including AutoCAD Civil settings, styles, label styles, tables, description keys, and point import/export formats) or the Prospector tree (including any AutoCAD Civil object, such as point groups).

Templates are listed under the Drawing Templates collection in the Prospector tree Master View.

The Drawing Template File Location path on the Files tab of the Options dialog box specifies where AutoCAD Civil templates are located.

**To use drawing templates**

1  In the Toolspace Prospector, click Master View from the drop-down list.
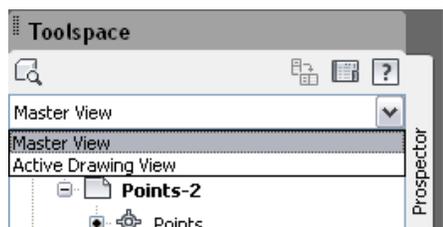


2  Expand the Drawing Templates collection and right-click a template to:
    ■  Create a new drawing based on the template.

■ Open the template.

## Out-of-the-box Templates

AutoCAD Civil drawing templates are included in the product.

The AutoCAD Civil drawing templates are based on the National CAD Standard version 3.1. The following templates are included:

■ _AutoCAD Civil 3D (Imperial) NCS

■ _AutoCAD Civil 3D (Metric) NCS

These templates contain an extended set of styles and settings.

**NOTE** These templates do not contain styles named 'Standard.' If you base a new drawing on a non-Civil 3D drawing template, such as acad.dwt, then styles named 'Standard' are created by default in the new drawing.

When you start a new drawing using the New command on the Application menu, you can choose which template you want to use.

The QNEW command, which is available on the Quick Access Toolbar, automatically uses the template specified in the AutoCAD Options dialog box. For more information, see "Select a Template When You Create a New Drawing" in AutoCAD Help.

### Drawing Template Layer Naming Conventions

The layers created in the AutoCAD Civil 2010 templates follow the National CAD Standards Rules (NCS). For more information about the National CAD Standards layering settings, see *http://www.nationalcadstandard.org*.

The layers follow the NCS standards as follows, with each element separated by a dash:

**<Discipline Designator> - <Major Group> - <Minor Group> - <Status>**

■ Discipline Designator: Required; the AutoCAD Civil templates use the C and V discipline designators, which stand for Civil and Survey/Mapping.

■ Major Group: Required; identifies elements such as roads, topographic elements, and storm sewers. To adhere to the standards, custom Major Group fields are not allowed.

■ Minor Group: Optional; identifies sub-elements such as road profiles. You can include up to two minor groups per layer name, and you can define your own custom Minor Groups. For example, the layer C-ROAD-LINE-EXTN has two Minor Groups: "Line" and "Extn".

■ Status: Optional; identifies phases of work.

For example, C-TOPO-MINR-N stands for *Civil - Topographic element - Minor Contours - New*.

### Plan Production Drawing Templates

Drawing templates for Plan Production tools are inlcuded in the Plan Prodution directory.

For information on using the Plan Production tools, see Plan Production Tools (page 1691).

## Layer Standards in Templates

Maintaining a standard for the use of layers in your drawing is easier if you have a drawing template with predefined layers, and styles that reference those layers when objects are created.

For consistency, save your template as an AutoCAD standards file (.dws format). Then, in the template, enter the STANDARDS command and in the Configure Settings dialog box, link the standards file to the template. If you want to enforce the standards, you can click the Settings button and activate notifications of standards violations, or override such violations. See AutoCAD Help for more information about the use of standards files.

For more information about layers, see Layers (page 58).

## Object and Label Styles in Templates

Consider designing styles to reflect the information requirements at different times in the project.

For example, create styles for objects in the design phase, and other styles for their final presentation. Also, you can create styles for different users of the drawings, showing object details and labels that present the data each type of user needs.

The styles for labels are another template item that should be developed with a top-down approach, similar to drawing settings and feature settings. At the drawing level, click Edit Label Style Defaults. In this context, you can set global standards such as those for text font, size, and label behavior around curves. As with the object styles, after setting label style defaults you can review the feature-level settings and apply any required overrides.

Ideally, your drawing template will have the perfect set of styles for your projects, but this usually takes some time to evolve. To assist the process, create a demonstration drawing that contains one or more surface, alignment, and other object types. Or you can explore the Sample_styles.dwg, that is supplied with AutoCAD Civil. After a default installation, you can find this file at: C:\Program Files\AutoCAD Civil 2010\Help\Civil Best Practices Guide\Sample_styles.dwg. This provides a good laboratory in which to display and develop styles. Consider designing styles to reflect the information requirements at different times in the project. For example, create styles for objects in the design phase, and other styles for their final presentation. Also, you can create styles for different users of the drawings, showing object details and labels that present the data each type of user needs.

For more information on styles, see Object Styles (page 53).

## Saving a Drawing as a Template

Create a new template from an existing drawing.

**To create a drawing template file from an existing drawing**

1   Click ![icon] ➤ Open.

2   In the Select File dialog box, select the file you want to use as a template.

3   Click OK.

4   Click ![icon] ➤ Save As.

5   In the Save Drawing As dialog box, under Files of Type, select the Drawing Template file type.

    DWT files must be saved in the current drawing file format. To create a DWT file in a previous format, save the file in the desired DWG format, and then rename the DWG file using a DWT extension.

6   In the File Name box, enter a name for the template.

7   Click Save.

**8** Enter a description of the template.

**9** Click OK.

### Quick Reference

Menu

 ➤ Save As

Command Line

SaveAs

# Working with Drawing Settings

Use drawing templates to eliminate duplication of effort and to help maintain consistency across drawings.

As a basis for configuring your drawing settings, you can choose an existing template that is closest to your needs. For more information about drawing templates, see Drawing Templates (page 109).

To see the available templates, click Application Menu ➤ New. If you want some preset object styles, the templates that have names beginning with "_AutoCAD Civil 3D", are recommended.

After choosing your initial template, the next logical step is to review and edit settings. On the Toolspace Settings tab, right-click the drawing name and click Edit Drawing Settings. The Drawing Settings dialog box includes five tabs and many settings.
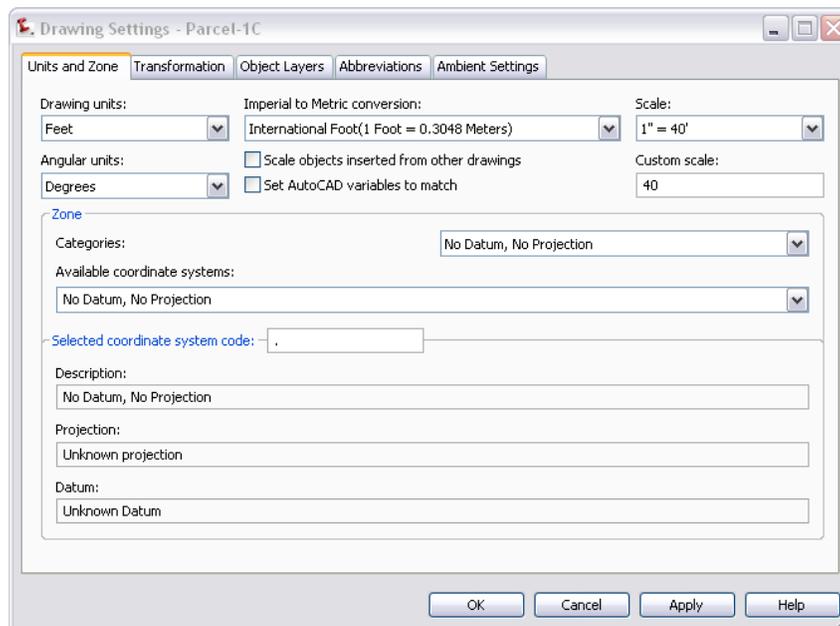


**Drawing Settings dialog box**

**NOTE** For detailed reference information about the Drawing Settings tabs, see

- The Units and Zones tab is used to set linear and angular units, intended plot scale, and coordinate zone for the current drawing.

- The Transformation tab provides functionality to transform the coordinate system specified on the Units And Zones tab to local specifications.

- The Object Layers tab provides a convenient table for assigning objects to layers, and creating or renaming layers. Objects on the same layer are turned on and off together if you change the layer visibility. For maximum display flexibility, spread objects and their components across several layers. The color of a layer affects all objects assigned to that layer and drawn in a style that uses the ByLayer color setting. If you set up styles this way, you can change the color of many objects by simply changing the layer color.

- The Abbreviations tab is a central location in which to set the abbreviations used in object labels, especially for alignment and profile geometry points.

- The Ambient Settings tab provides global default settings, such as the precision of numeric values, and the visibility of tooltips.
One setting on this tab often overlooked is the Save Command Changes To Settings under the General setting group. If you set this to Yes, then whenever you change a setting, such as the radius of an alignment curve, it is saved as the default value for next time.

  Another important setting is Independent Layer On. Set it to No if you want to be able to control object visibility by turning layers on and off.

  The ambient settings you configured at the drawing level are available at the feature level in case you want to override them.

  On the Ambient Settings tab, you see an arrow in the Child Override column next to any setting that is changed for one or more features (objects) in the drawing. You can cancel the override by clicking the arrow, and you can click the lock icon to lock any setting and prevent overrides at the feature level.
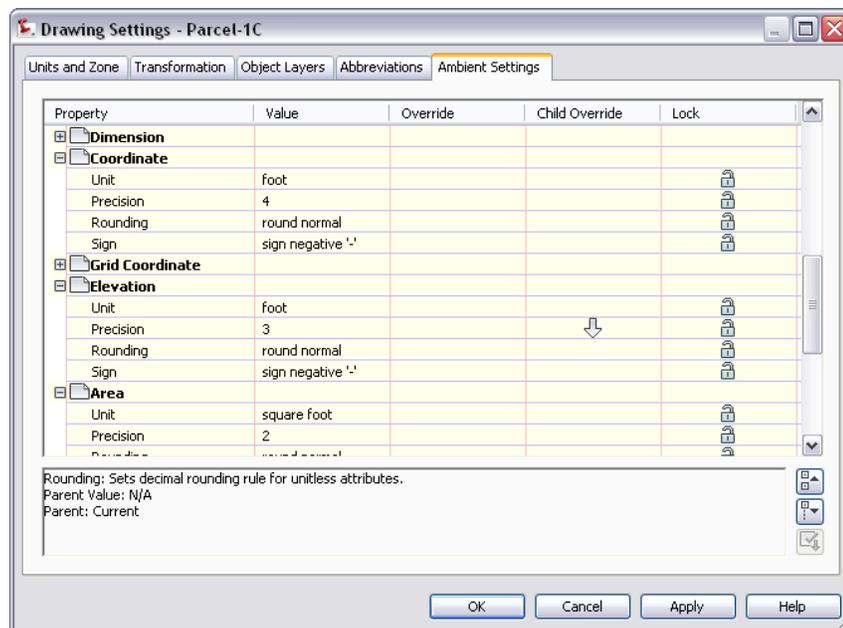


**Use Ambient Settings to cancel style overrides.**

After configuring the drawing settings, you should work down through Point, Surface, and other features in the Settings tree, right-clicking each feature and selecting Edit Feature Settings. Doing this allows you to examine the default styles for each feature and its labels, the feature name format, and other settings that you may want to modify.

For more information about AutoCAD Civil drawing settings, see Specifying Drawing Settings (page 63).

# Creating a New Drawing

Create a new drawing using standard AutoCAD commands.

**To create a new drawing**

**1** Click  ➤ New.

**2** In the Select Template dialog box, select a template on which the drawing will be based, and click Open.

## Quick Reference

Menu

 ➤ New

Toolspace

Prospector (Master View): Drawing Templates ➤ right-click <template name> ➤ Create New Drawing

Command Line

New

# Opening an Existing Drawing

Open an existing drawing using standard AutoCAD commands.

If more than one drawing is open, the name of the active drawing is displayed in boldface in the Prospector tree.

**NOTE** You can change the active drawing by right-clicking a drawing name in the Prospector tree Master View and clicking Switch To.

**To open an existing drawing**

**1** Click  ➤ Open.

**2** In the Select File dialog box, browse to the folder containing the drawing, select the drawing, and click Open.

Menu

 ➤ Open

Command Line

Open

# Working with Drawing Data

AutoCAD Civil provides many tools and features to make it easier to work with drawing data.

For information on other AutoCAD Civil utilities, see Utilities (page 1603).

# Specifying Locations in a Drawing

When you are prompted for a location in a drawing, you can use one of several methods to specify the coordinates.

### Select an Existing Point

Use the mouse to select an AutoCAD Civil point in the active drawing.

### Specify a Location in the Drawing

Enter X, Y (and Z, if prompted) coordinates at the command line, separated by a comma. Alternatively, click a location in the drawing. You can use AutoCAD Object Snaps to select a point accurately.

### Use Civil Transparent Commands

Some of the Civil transparent commands calculate point locations from angles, distances, and drawing geometry. For more information, see Transparent Commands (page 1581).

### Use the Coordinate Tracker

You can send drawing coordinate information to the command line based on cursor location and a specified surface. For more information, see (page 1623).

# Selecting Similar Objects

Use the Select Similar command to select all objects that have the same style and that are on the same layer.

**To create a selection set of objects that have the same style and that are on the same layer**

1 Select the first object in the selection set.

2 Right-click, and click Select Similar.

### Quick Reference

Ribbon

Click <object name> tab ➤ General Tools panel ➤ ▼ ➤  Select Similar

Object Shortcut Menu

　Edit menu ➤ Select Similar

Command Line

　SelectSimilar

# Isolating Objects

AutoCAD Civil provides an easy-to-use feature for temporarily hiding and displaying selected objects in your drawing.

Use this operation when you want to focus on individual areas of large drawings and need to display or hide some of the objects.

Do the following:

- Create an object selection set to hide or display.

- Fine-tune the selection set by adding objects.

- Save a drawing with the visibility settings you have selected.

## Isolating Objects to Display

Isolate objects that you want to display in a drawing. This is preferable when you have a large drawing with many objects and need to display only a small number of objects.

Run the Isolate Objects command either from the shortcut menu of an object or from the AutoCAD Civil drawing window status bar.



**Isolate Objects command on the drawing window status bar**

**To isolate objects and display them in a drawing**

1　Select the objects in the drawing that you want to isolate and display.

2　Right-click, and click Isolate Objects ➤ Isolate Selected Objects.

Alternatively, you can click 💡 on the right side of the drawing window status bar, then click Isolate Objects, and select the objects you want to isolate.

When you have isolated objects in your drawing, the light bulb in the drawing window status bar turns red 💡 .

3　If you want to isolate additional objects in your selection set, select one or more isolated objects, and run the Isolate Objects command again.

Alternatively, you can click the red light bulb 💡 in the drawing window status bar and click Isolate Additional Objects.

When you end the object isolation, as described in Ending Object Isolation in a Drawing (page 118), all objects are displayed.

## Quick Reference

Ribbon

Click <object name> tab ➤ General Tools panel ➤ Isolate Objects 

Command Line

IsolateObjects

# Isolating Objects to Hide

Isolate objects that you want to hide in a drawing. This is preferable when you want to display a large number of objects and hide only a few of them.

**To isolate objects and hide them in a drawing**

1 Select the objects in the drawing that you want to hide.

2 Right-click, and click Isolate Objects ➤ Hide Selected Objects.

The objects you selected are now hidden in the drawing.

Alternatively, you can click the light bulb  in the drawing window status bar and click Hide Objects.

---

**NOTE** After isolating and hiding objects multiple times, it may be difficult to know whether any objects are hidden, or whether all objects are displayed in the drawing. When you have isolated objects in your drawing, the light bulb on the right side of the drawing window status bar is displayed in red  . When all objects are displayed and none are isolated or hidden, the light bulb is displayed in yellow  . Alternatively, to ascertain that no objects are hidden, select an object in the drawing, right-click, and click Isolate Objects. If the End Object Isolation command is unavailable, all objects are displayed. If the shortcut menu command End Object Isolation is active, there are hidden objects in the drawing. To display all objects, see Ending Object Isolation in a Drawing (page 118).

---

## Quick Reference

Ribbon

Click <object name> tab ➤ General Tools panel ➤ Hide Objects

Command Line

HideObjects

## Ending Object Isolation in a Drawing

End the object isolation in a drawing and display all hidden objects.

**To end object isolation in a drawing and display all hidden objects**

1   Select an object in the drawing.

2   Right-click and click Isolate Objects ➤ End Object Isolation.

Alternatively, click 💡 on the right side of the drawing window status bar. Click End Object Isolation.

**NOTE** If no objects have been isolated, the End Object Isolation command is unavailable and the light bulb in the drawing window status bar is displayed in yellow 💡 . In that case, no further action is necessary.

### Quick Reference

Ribbon

Click <object name> tab ➤ General Tools panel ➤ End Isolation

Command Line

UnisolateObjects

## Saving Isolated Objects

Save a drawing in which some objects are isolated for viewing while others are hidden. The next time you open the drawing, it is displayed with the isolation settings you defined.

For example, you could save a specific view setting to distribute the file to a contractor who needs to see only a specific part of the drawing. You could also save a specific view setting in a drawing that is used as an external reference within another drawing.

To save a drawing with isolated objects, create the appropriate object selection set as described in Isolating Objects to Display (page 116) and Isolating Objects to Hide (page 117). Save the drawing. The isolation settings are saved in the drawing and are displayed accordingly when you reopen the drawing.

**NOTE**  If you want to distribute a drawing with isolated objects, make sure that you and the recipient of the drawing are working with the same version of AutoCAD Civil. A recipient who is working with an earlier version of AutoCAD Civil, AutoCAD, or Object Enabler is not able to display the hidden objects. In this case, end the object isolation before saving and distributing the drawing.

## Isolating Objects in External References

When you are working with external references, you can hide and isolate objects. Consider the following points:

■   You can isolate an external reference within a host drawing as a whole by selecting it and isolating at as described in Isolating Objects to Display (page 116).

■   You can isolate individual objects within an external reference by opening the external reference, isolating and hiding objects as needed, saving the external reference drawing, and then reloading the external reference.

- You cannot isolate or hide objects in an external reference while editing it with REFEDIT in the host drawing. You need to use XOPEN and save the desired Hide/Isolate settings within the external reference.

- You cannot display hidden objects in an external reference while editing it with REFEDIT in the host drawing. You need to use XOPEN and save the desired Hide/Isolate settings within the external reference.

# Dragging Items Within or Between Drawings

Copy styles, description keys, and pipe rules using the Settings tree drag-and-drop capability.

You can drag items from the Settings tree into the drawing window to copy them, or you can drag them onto a drawing icon in the Settings tree.

### Dragging Items between Drawings or Templates

You can copy the following items by dragging them from one drawing or drawing template to another:

- Styles, including label and table styles

- Description keys

- Pipe rules

Both the item you want to copy and the name of the destination drawing must be visible in the Settings tree before you begin the drag-and-drop operation.

To drag an item from one drawing to another, click the item or collection you want to copy, hold down the left mouse button, and drag the item to the name of destination drawing in the Settings tree. When the cursor changes to     , release the mouse button to copy the item into the drawing. The item is automatically added to the correct collection in the drawing.

If the drawing already contains an item with the name of the item being copied, the Duplicate Item Name Dialog Box (page 1812) is displayed which allows you to decide how to handle the conflict.

If you drag a label style with one or more child styles from one drawing to another, only the selected style is copied into the drawing; the child styles are not copied. If you copy a child style into another drawing, the parent style is also copied.

### Dragging Items within a Drawing

You can copy label styles by dragging them from one level in a label style collection to another within a drawing, as follows:

You can drag and drop multiple styles from one drawing to another at the same time.

- Drag a child label style (and its children) up to a higher level in the style collection to sever the parent/child relationship.

- Return the child label style to its original location in the tree to reinstate the parent/child relationship.

- Drag a child label style to another label style collection. The child label style inherits the properties of the new parent.

Both the label style you want to copy and the destination location of the style must be visible in the Settings tree before you begin the drag-and-drop operation.

To drag a label style within a drawing, click the label style you want to copy, hold down the left mouse button, and drag the label style to the destination collection. When the cursor changes to ![cursor], release the mouse button to drop the style into the collection located at the tip of the cursor arrow.

When you copy a label style within a drawing, all child styles are copied with the parent style.

# Consistent Editing Methods

Object editing in CEV_ProdName_CEV uses an approach that is consistent for all objects. The main editing tools are described here.

## Item View

When you click an object or an object collection on the Prospector tab, for example, Points or Alignments, an item view appears. An item view can be either a list view or a graphical view, depending on the object selected.

The item view presents a table in which you can review and edit data for each object in the selected collection. For example, if you select a point group, the item view table includes a row for each point in the group.
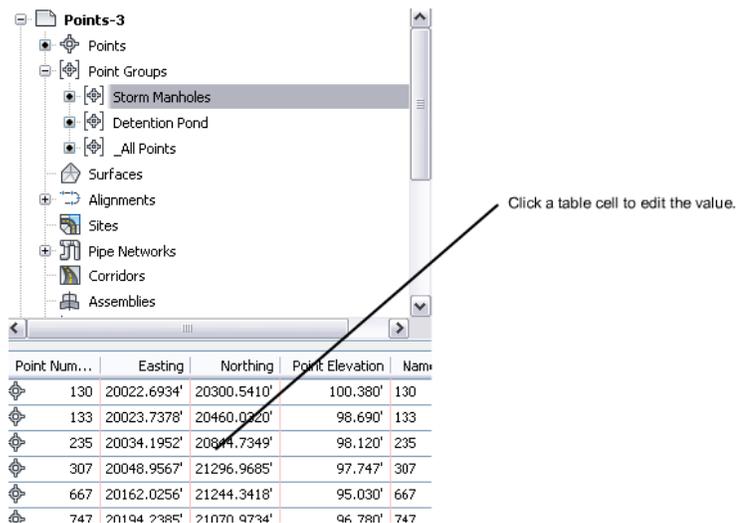


**Item view showing a group of points**

## Grips

When you select an object in the drawing, grips appear on the object. You can use these grips to click and drag the object to a new location.

For example, when editing alignments, you can use grips to move points of intersection or points of line-arc tangency.
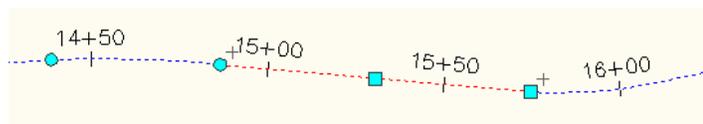


**Alignment grips for manual editing**

When you use a grip to drag a label object to a new location, a round reset grip appears. You can click this round grip to return the label to its original location. If there are multiple leader lines, click the round grip once to restore the leader to its orignal state. Click the round grip twice to restore a dragged label to its undragged state.
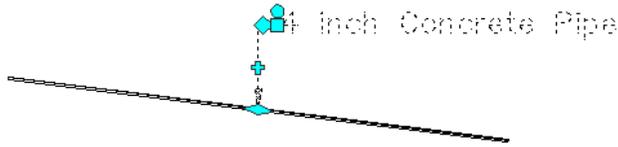


**Use round grip to to restore label leader or to reset label**

## Panorama Window

Some object types, such as alignments and profiles, use the Panorama window to display a table of entities that make up that object.

The Panorama window is a floating, dockable window that you can keep open as you work. It can include several tables, called vistas, on different tabs. Panorama data shown in black text can be edited; data shown in gray text cannot be edited.
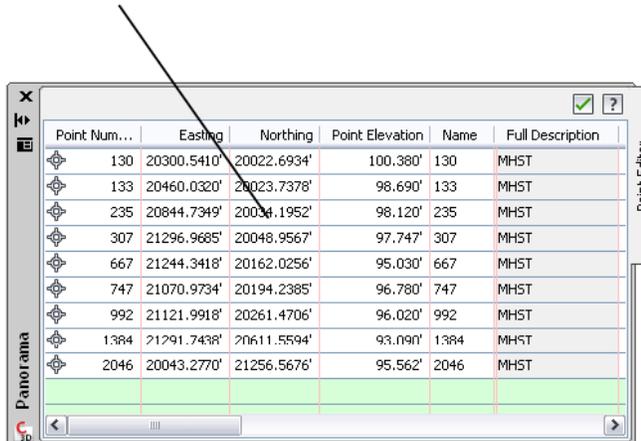


**Panorama window showing an alignment**

## Property Editors

When you right-click any object on the Prospector tab, and then click Properties, you can view all the AutoCAD Civil properties of that object, and edit some of them.

These properties typically include the styles, labels, related objects, and some structural details of the current object.
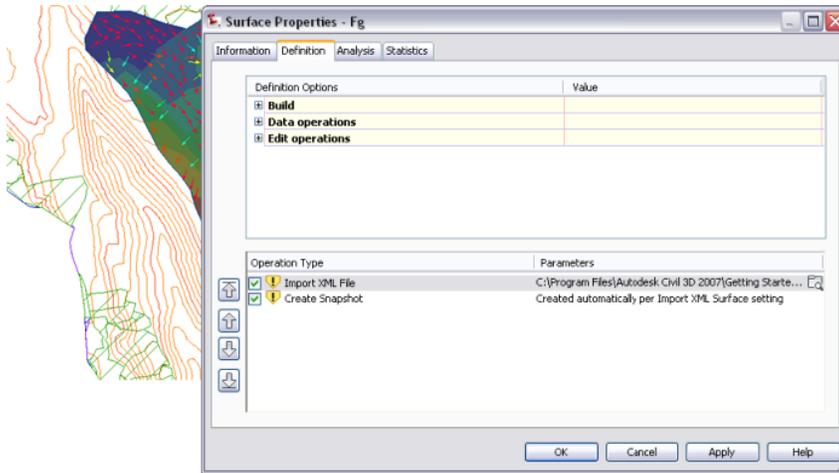
**Properties editor showing surface properties**

# AutoCAD Properties Editor

When you right-click an object in the drawing, and then click Properties, the AutoCAD Properties dialog box is displayed.

You can use this dialog box to review and edit AutoCAD properties, such as the layer on which the object is drawn. You can view the style assigned to the object, and can also change it here. You can also edit an object style on the Settings tab of Toolspace. Right-click the object style. Click Edit to open the <feature> Style dialog box.
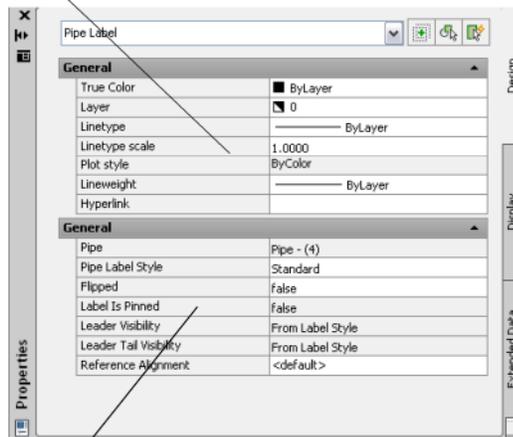


**Editing AutoCAD properties for a pipe label**

# Tracking Stations in Alignments, Profiles, and Sections

The Station Tracker provides visual cues to help you locate the stations that match your cursor position when working with alignments, profiles and sections.

■ When the cursor is above a horizontal alignment, and one or more profile views for the same horizontal alignment are in the drawing, a temporary line is drawn in each profile view at the station defined by the orthogonal projection of the cursor onto the horizontal alignment. The temporary line is drawn to the full height of the profile view, and moves as the cursor moves.

■ When the cursor is within a profile view, a temporary line is drawn orthogonal to the horizontal alignment at the cursor's station. The line extends 200 world distance units to the left and right sides of the horizontal alignment. As the cursor's position changes in the profile view, the temporary line's position is redrawn at the new station location.

■ When the cursor is within a section view, a temporary line is drawn orthogonal to the horizontal alignment at the station of the cross-section view. The line extends 200 world distance units to the left and right sides of the horizontal alignment. Additionally, a vertical line is drawn in each associated profile view. The vertical line is drawn at the station of the section view.

This setting is controlled per drawing.

For information on the use of the similar station tracker utility in corridor section views, see Tracking Corridor Sections at Stations (page 1364).

**To display the Station Tracker in the current viewport**

➤ Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ Current Viewport  .

**To display the Station Tracker in all viewports**

➤ Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ All Viewports  .

**To turn off the display of the Station Tracker**

➤ Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ Off  .

## Quick Reference

Ribbon

Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ Current Viewport 

Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ Off 

Click Analyze tab ➤ Inquiry panel ➤ Station Tracker drop-down ➤ Off 

Command Line

ShowDrawingTips
ShowDrawingTipsFull
HideDrawingTips

# Drawing Command Reference

You can use commands to quickly access drawing functionality.

The following table lists the drawing-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| HideObjects | Selects objects to hide in a drawing (page 117) |
| IsolateObjects | Selects objects to display in a drawing (page 116) |
| New | Creates a new drawing (page 114) |
| Open | Opens an existing drawing (page 114) |
| SelectSimilar | Selects all objects that have the same style and that are on the same layer (page 115) |
| UnisolateObjects | Ends object isolation in a drawing and displays all hidden objects (page 118) |
| ShowDrawingTips | Displays the Station Tracker in the current viewport (page 123) |
| ShowDrawingTipsFull | Displays the Station Tracker in all viewports (page 123) |
| HideDrawingTips | Turns off the display of the Station Tracker (page 123) |

# Project Management

# 7

AutoCAD Civil offers a range of options for managing engineering projects. Once you understand these options, you can determine the best data structure for your projects.

For information on recommended workflows for project management, see the *AutoCAD Civil Workflow Guide*.

## Understanding Project Management

Depending on project complexity, your project management system can range from a few drawings in a folder to large database with shared objects and security controls.

When deciding on a structure to use for managing your projects, consider these factors:

■  The number and complexity of drawings

■  The size of the project team and number of people requiring access to the files

■  The relative need for ease of use versus data security

For more information on project management options and the decision-making process, see Project Management in the *AutoCAD Civil Best Practices Guide*.

## Object Sharing

Sharing design objects such as surfaces and alignments across drawings can provide major savings in project time lines, if done correctly.

Three main methods are available for sharing data within a project. These are also known as data reference types:

■  External references (xrefs), in which the contents of one drawing (DWG) is imported into another.

■  Data shortcuts between DWGs

■  Object references in Autodesk Vault

For a detailed description of each method, including advantages, limitations, and best practices, see Project Management in the *AutoCAD Civil Best Practices Guide*.

A best practice common to all project structures is to organize individual design objects such as alignments and surfaces in separate drawings. These objects are referenced into base drawings during the design process. Complete engineering plans and final production drawings are prepared with a combination of object references and drawing references.
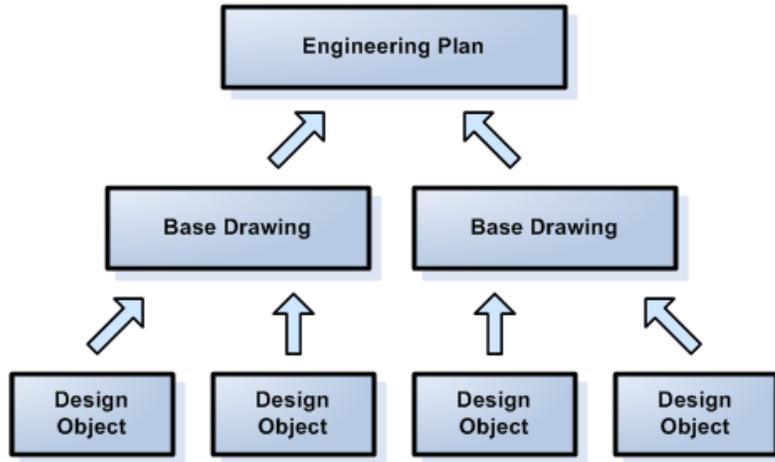


## Project Management on the Prospector Tree

Projects can be displayed and managed from collections on the Prospector tree.
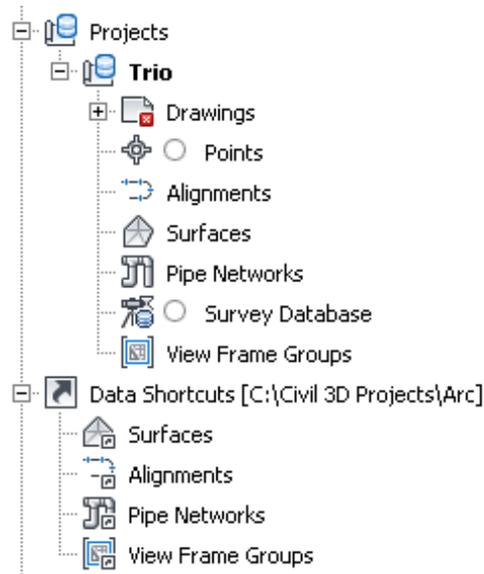


**Prospector collections for projects**

The figure shows the collections for a Vault project named Trio and a data shortcut project named Arc. Both project types are created from the default project template. As a result, they have identical collections of surfaces, alignments and other objects that can be referenced into multiple drawings.

You can store multiple projects in the same working folder. The default location of the working folder for data shortcuts projects is C:\Civil Projects, and for Vault projects is My Documents\Vault (Microsoft Windows XP), or Documents\Vault (Microsoft Vista).

Several Vault projects can be displayed in the Projects collection. In the figure, you see two projects, Trio and Vast. For data shortcut projects, you switch from one to another by changing the data shortcuts folder.

Most commands for managing projects are accessible by right-clicking specific objects in the project collections on the Prospector tree.

**See also:**

- Using Data Shortcuts (page 127)
- Using Vault (page 138)

# Transition from Land Desktop

Compared to the standard format used in AutoCAD Land Desktop, project structure within AutoCAD Civil can be customized to suit your requirements.

AutoCAD Civil includes an Import Data From Land Desktop command. This command makes use of the following dialog box, which enables you to selectively bring objects into new drawings for use in AutoCAD Civil. For more information, see Using the Import Data From Land Desktop Command.
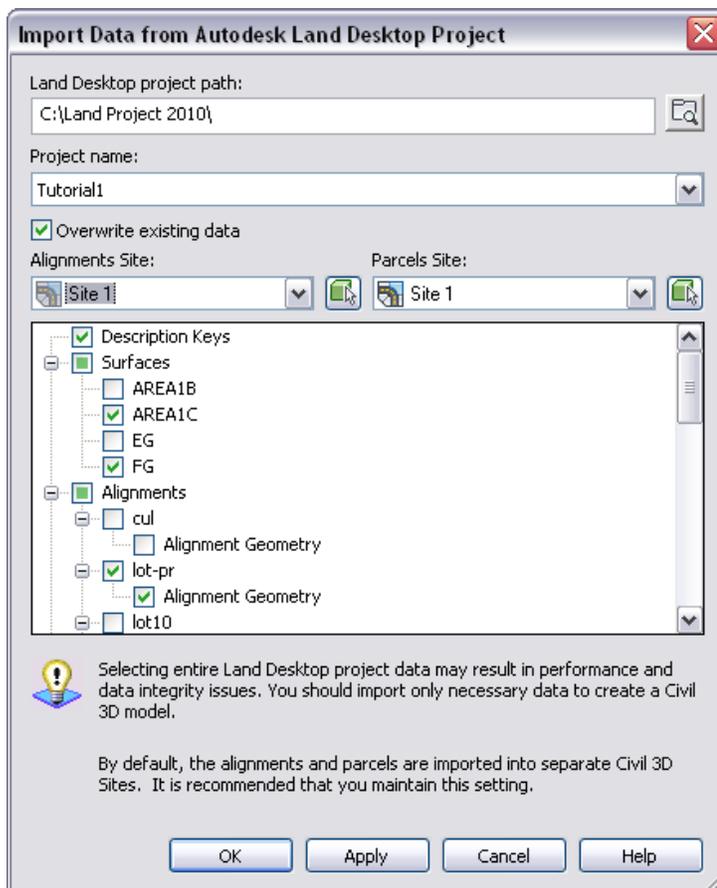


The recommended use of this Import function is to bring individual surfaces, alignments, or pipe runs into new drawings, one object per drawing, for use in AutoCAD Civil projects.

# Using Data Shortcuts

This section explains the concepts and procedures for managing data shortcuts.

Using data shortcuts involves two main processes:

■ Create data shortcuts from their source drawings.

■ Create data references in consumer drawings to source objects in source drawings.

Data shortcuts can be used for surfaces, alignments, profiles, pipe networks, and view frame groups.

If you create a reference to a profile data shortcut, a data shortcut to the parent alignment is also created.

### Relationship Between Shortcuts and Data References

A data shortcut provides a direct path to the location of a shareable source object, either a surface, alignment, profile, pipe network, or view frame group. In a single operation, you can create data shortcuts for multiple objects in a source drawing. The data shortcut is used only for the creation of data references.

A data reference is a read-only copy of a source object, inserted into another drawing, often called a consumer drawing. From the Prospector tab, you can select a shortcut for an object and create a reference to that object in the active drawing. The data reference maintains an active link to the source object in the source drawing, without relying on the data shortcut.

**NOTE** The term 'project' is used here for ease of reference, and to explain a sample data structure. Formal projects and particular folder structures are not required with data shortcuts and references, but some form of structure usually helps to keep related data organized.

On the Toolspace Prospector tab, the Data Shortcuts node displays the path to the active project, where data shortcuts are stored. These data shortcuts can be created from any drawings associated with the active project. The figure shows how the Data Shortcuts node appears if the active project is Arc.



**Shortcut folders on the Prospector tree for sample project Arc**

If you change the active project, the new project name appears on the Data Shortcuts node, and the subfolders contain only the shortcuts in the new active project. A particular consumer drawing can contain data references from multiple projects.

### Data Reference Attributes

The reference object has read-only geometry, but the user can apply a local object style, apply custom annotation, do limited analysis, and access the source object's properties. The reference object takes up less file space when the consumer drawing is saved. If the source object is edited, the synchronization process ensures that changes are reflected in the reference object. Synchronization occurs when the consumer drawing is opened, and can also be done manually by a user. You can promote a data reference to an independent object that can be edited; after promotion it is no longer updated from its original source object.

Data Shortcut Best Practices

# Project Setup

This section explains how to organize the data for a new project, including some configuration options to consider.

## Setting a New Working Folder

Set a new working folder as the location to store data shortcut projects.

The default working folder for data shortcut projects is C:\Civil Projects. The default working folder is also used for Autodesk Vault projects and Survey projects.

If you work with Autodesk Vault, Survey, and data shortcut projects, you should have separate working folders for each project type, for ease of management.

**To set a new working folder**

1    In Toolspace, on the Prospector tab, right-click Data Shortcuts ➤ Set Working Folder.

2    Navigate to the desired folder and select it.

### Quick Reference

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ Set Working Folder 🗂

Command Line

SetWorkingFolder

## Creating a Data Shortcut Project Template

Create a new template if the sample template is not suitable.

To create your own project template, copy the default template, _Sample Project and save it with a new name. To maintain the automatic storage of new data shortcut XML files, do not change the _Shortcuts folder and its subfolders. Your new project template will appear on the list of templates when you create a new data shortcut folder.

**To create a new template for data shortcut projects**

1    Navigate to the folder C:\Civil Project Templates.

2    Copy the folder _Sample Project, then paste it in the same location and rename it. This is your new template.

3    Open your new template and review the structure of folders and subfolders.

4    Do not change the top-level folder, _Shortcuts, or any folders inside it, but feel free to change the names of other folders and their hierarchical structure.

## Creating a New Shortcuts Folder

Create a new folder for storing a set of related project drawings and data shortcuts.

Create a folder name, which should reflect the project name, and specify whether to use a project template to organize your data.

The figures show the structure created by the default project template.

**Shortcut project template: the Prospector tree icons**
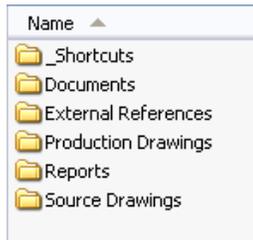


**Shortcut project template: the working folder**

Using a project template is a best practice that you will appreciate as your collection of projects and data expands.

**To create a new data shortcut folder**

1  In Toolspace, on the Prospector tab, right-click Data Shortcuts ➤ New Data Shortcuts Folder.

2  In the New Data Shortcut Folder dialog box, specify a name, and optionally a text description of the project folder.

3  Optionally, select Use Project Template, browse to another location for the project template folder, and click the name of a project template.

4  Click OK. The new project folder is created in the working folder.

### Quick Reference

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ New Shortcuts Folder 

Command Line

NewShortcutsFolder

## Setting the Data Shortcuts Folder

Specify the project path for data shortcuts.

The path to the current data shortcuts folder (also known as the project folder) is specified on the Data Shortcuts node in Toolspace, on the Prospector tab.

The project folder typically contains both the data shortcuts and the source objects for data references.

Data references that you create in the active drawing come from the active project folder.

Setting or changing the data shortcuts folder reloads the shortcuts into all drawings in that folder. You can reload the shortcuts to apply changes made in the Data Shortcut Editor window. See Editing Data Shortcuts (page 137).

**To set the data shortcuts folder**

1  In Toolspace, on the Prospector tab, right-click Data Shortcuts and click Set Data Shortcuts Folder.

2  In the Set Data Shortcuts dialog box, click the name of the project you want to work with.

3  Check that the new project path is displayed on the Data Shortcuts node on the Prospector tab.

## Quick Reference

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ Set Shortcuts Folder ⬚

Command Line

SetShortcutsFolder

Toolspace Data Shortcut Collection

Prospector tab ➤ Data Shortcuts ➤ Set Data Shortcuts Folder

Dialog Box

Set Data Shortcuts Folder (page 2208)

# Creating Data Shortcuts

Create data shortcuts from the active drawing.

Data shortcuts are stored in the _Shortcuts folder for the active project and used to create data references to the source objects in other drawings.

Each data shortcut is stored in a separate XML file.

**NOTE** The system will not allow you to create multiple shortcuts for a given object or to create a data shortcut to a reference object from another drawing.

**To create data shortcuts**

1  In Toolspace, on the Prospector tab, right-click Data Shortcuts ➤ Create Data Shortcuts.

2  In the Create Data Shortcuts dialog box, click any objects to be referenced in other drawings, and then click OK.

## Quick Reference

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ Create Data Shortcuts ⬚

Toolspace Data Shortcut Collection

Prospector tab ➤ Data Shortcuts ➤ Create Data Shortcuts

Menu

General ➤ Data Shortcuts ➤ Create Data Shortcuts

Command Line

CreateDataShortcuts

Dialog Box

Create Data Shortcuts (page 2200)

# Referencing a Project Object

Import a reference object into the active drawing.

A project object can be referenced into the active drawing, which is called the consumer drawing.

Before referencing a project object, the current drawing must be saved.

Only one copy of a reference object can be imported into a consumer drawing. If a reference copy exists, the referencing menu option is not available.

Objects can be referenced only from the active project, which is listed in Toolspace, on the Prospector tab.

**See also:**

■  Setting the Data Shortcuts Folder (page 130)

**To reference a data shortcut**

1   In Toolspace, on the Prospector tab, expand the Data Shortcuts collection to find the object you want to reference.

2   Right-click the object. Click Create Reference. A copy of the object is inserted into the active drawing.

### Quick Reference

Toolspace Data Shortcut Collection

Prospector tab ➤ Data Shortcuts, Right-click object name ➤ Create Reference

# Removing a Data Shortcut

Remove a data shortcut to make it unavailable for future data references.

You can remove a data shortcut from the Data Shortcuts collection in Toolspace, on the Prospector tab. Doing this does not remove any data references to the source object that exist in drawings, but it prevents anyone from creating more references to the object.

**To remove a data shortcut**

1   In Toolspace, on the Prospector tab, expand the Data Shortcuts collection to find the shortcut you want to remove.

2   Right-click the object and click Remove.

# Enabling and Disabling the Data Shortcuts Node

Control access to the use of data shortcuts.

You can remove the Data Shortcuts node from the Prospector tab of Toolspace. This can be useful if you want to prevent the use of data shortcuts.

Use the ShortCutNode command to either hide (disable) or show (enable) the Data Shortcuts node.

**To control access to the Data Shortcuts node**

1 On the command line, enter shortcutnode.

2 Set the value to 0 to disable the Data Shortcuts node, or 1 to enable the node.

# Synchronizing References

Synchronizing a data reference brings in any changes made to its source object.

When a drawing contains references to data shortcuts, the  icon is displayed at the bottom-right side of the window:

Data references are synchronized automatically when the drawing is opened. Otherwise, flags appear in the Prospector tree and the status bar of the drawing window when a reference object is out of date.



When references become out of date with the data shortcuts, the  icon displays a message prompting you to synchronize the references. Additionally, in the Prospector tree, the  icon is displayed next to reference objects that need to be synchronized.

**To synchronize a reference object**

1 In Toolspace, on the Prospector tab, expand the drawing node. Expand the object collection (for example, Surfaces).

2 Right-click the out-of-date reference object. Click Synchronize.

> **NOTE** The Synchronize menu item is displayed only for reference objects that are out of date.

**Quick Reference**

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ Synchronize References 

Toolspace Shortcut Collection

Prospector tab: <drawing-name> ➤ <object-collection> ➤ Right-click <object-name> Synchronize

Command Line

SynchronizeReferences

Drawing

Right-click 🖼 ➤ Synchronize

# Saving a Data Shortcut Project to Vault

Copy an entire project to a new project in Autodesk Vault, preserving all object references.

You can use the Add to New Vault Project command on the Data Shortcuts node of the Toolspace Prospector tab to copy a project to Vault. For an alternative method, see Importing a Project (page 154).

### Usage Guidelines

- Before using this command, you must be logged in to Vault, and the project you want to add to Vault must appear on the Prospector Data Shortcuts node as the current project.

- As a best practice, put all project drawings and files within a project folder, using a standard template. The structure of folders will be replicated in Vault, including the paths to all source drawings.

- The name you assign to the Vault project cannot be the same as the data shortcut project folder name.

- After being added to Vault, the data shortcut project remains in place and is usable.

### To copy a project to Autodesk Vault

1 On the Toolspace Prospector tab, right-click Data Shortcuts ➤ Add To New Vault Project.

2 In the New Project dialog box, enter a project name, and optionally a Description. This project name cannot be identical to the data shortcut project folder name.

3 Click OK. The Vault project is created, and appears on the Projects node on the Prospector tab.

### Quick Reference

Toolspace

Right-click Data Shortcuts ➤ Add To New Vault Project

# Promoting References

Promote references to independent objects.

A promoted reference becomes an independent object, and is no longer automatically updated if the original source is edited.

### To promote a reference

1 In Toolspace, on the Prospector tab, expand the Data Shortcuts collection to see the data shortcut of interest.

2 Right-click the reference object and then click Promote.

### To promote all references in a drawing

1 Manage tab ➤ Data Shortcuts panel ➤ Promote All Data References 🖼 .

2 In the Promote All Data References dialog box, click Promote All Referenced Objects Into The Current File.

### Quick Reference

Ribbon

Single object: Select the reference object. Click ➤ Name tab ➤ Modify panel ➤ Promote Data Reference 

All objects in drawing: Click Manage tab ➤ Data Shortcuts panel ➤ Promote All Data References 

Menu

Click General menu ➤ Data Shortcuts ➤ Promote All Data References

Command Line

PromoteAllReferences
PromoteReference (for surfaces, alignments, and profiles)
PromoteNetwork
PromoteViewFrameGroup

# Validating Data Shortcuts

Data shortcuts can become invalid if changes occur to the names or locations of drawings or reference objects.

When a data shortcut reference is broken, the  icon is displayed next to the invalid data reference in Toolspace, on the Prospector tab.

Validation reloads the data references if the drawing paths have not changed. Otherwise, you must repair broken references.

**To validate data shortcuts**

■   In Toolspace, on the Prospector tab, right-click Data Shortcuts ➤  Validate Data Shortcuts.
    The system checks data shortcut references. If any references are broken, the Event Viewer opens and displays details that you can use to fix the problem.

### Quick Reference

Ribbon

Click Manage tab ➤ Data Shortcuts panel ➤ Validate Data Shortcuts 

Command Line

ValidateShortcuts

# Repairing Broken References

You can recover from minor changes such as a moved or renamed drawing.

When a data shortcut reference is broken, the  icon is displayed next to the invalid data shortcut in Toolspace, on the Prospector tab for the host or consumer drawing.

Broken references typically occur if a source drawing has been renamed or moved to a different folder. If you can identify the new name or location, it is quite easy to repair the references. If the source drawing contains multiple data shortcuts, once you identify the drawing you are prompted to repair all of the references.

**To repair broken references**

**1** In Toolspace, on the Prospector tab, right-click the object name of the broken reference, and then click Repair Broken References.

**2** In the Choose the File dialog box, navigate to the source drawing for the referenced object, and then click Open.

If the source drawing contains other broken references, the Additional Broken References dialog is displayed. You can click the Repair All Broken References button, or click Cancel to repair only the broken reference you selected.

### Quick Reference

Toolspace

Right-click broken reference object ➤ Repair Broken References

Command Line

RepairBrokenReferences

## Accessing the Data Reference Source Drawing

Open the source drawing for a data reference.

This is useful if the source object requires editing.

**To access a data shortcut source drawing**

**1** In Toolspace, on the Prospector tab, expand the Data Shortcuts collection to see the data shortcut of interest.

**2** Right-click the data shortcut, and then click Open Source Drawing.

The source drawing for the data shortcuts is displayed in a new document window.

### Quick Reference

Menu

In Toolspace, on the Prospector tab, right-click Data Shortcut ➤ Open Source Drawing.

## Migrating Data Shortcuts

You can update data shortcuts to the current format or move them to a different folder.

Use the stand-alone Data Shortcuts Editor to save one or more data shortcuts to a specified folder. If the shortcuts were created with a previous version of AutoCAD Civil, they will be updated to the current format in this process.

**To migrate data shortcuts**

**1** From the Windows desktop, click the Start menu and navigate to your list of programs.

**2** Open the AutoCAD Civil 2010 folder and click Data Shortcuts Editor. The editor opens in a separate window.

**3** In the Data Shortcuts Editor window, click File menu ➤ Open Data Shortcuts Folder, and navigate to the shortcuts folder that contains the data shortcuts you want to edit. This folder is typically inside a named project folder.

**4** In the shortcut table, display the data shortcuts you want to migrate. Display all of them by clicking the data shortcut folder at the top of the tree, or display a subset by clicking an object type such as Surfaces or Alignments.

**5** Click File menu ➤ Save As, then specify the folder where you want to save the data shortcuts.

**6** Click OK to save and update the data shortcuts.

# Editing Data Shortcuts

Use the Data Shortcuts Editor for bulk operations on data shortcuts.

The Data Shortcuts Editor is a stand-alone application that you can use to edit multiple shortcut XML files safely and efficiently. For example, if you move an entire project from one working folder to another, you can use the Data Shortcuts Editor to update the paths to the data shortcut objects.
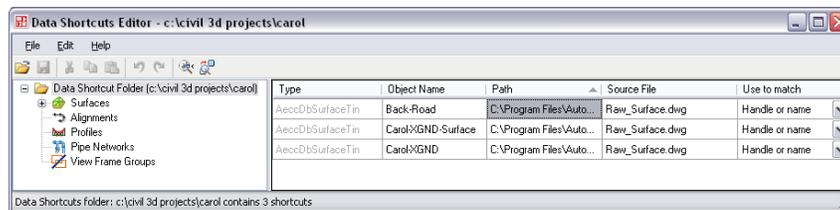


**The Data Shortcuts Editor window**

### Precautions

Before editing data shortcuts, ensure that no one else is currently creating or editing the same set of data shortcuts within AutoCAD Civil. Otherwise, one user can overwrite changes made by the other.

Changes made in the Data Shortcut Editor window do not take effect until shortcuts are reloaded. To reload the shortcuts, use the procedure Setting the Data Shortcuts Folder (page 130), or close AutoCAD Civil and restart it.

### Best Practices

On the Data Shortcut Editor's Edit menu, use the Find and Replace command to quickly change text strings in the names of data shortcuts, paths, or source drawings.

In the shortcut table, set the Use to Match column to associate data shortcuts with their source objects by means of the source object's name or handle, or both. The object's handle is its unique identifier used by the system. Users cannot change the handle, so it can be the most reliable matching attribute if an object's name has changed.

One situation in which you would match by name is if you delete an object, then re-create it and apply the same name. The new object has a different handle, but if you use the match-by-name option, the old data shortcuts will automatically reference the new object.

### See also:

■ Migrating Data Shortcuts (page 136)

### To access the Data Shortcuts Editor

**1** From the Windows desktop, click the Start menu and navigate to your list of programs.

**2** Open the AutoCAD Civil 2010 folder and click Data Shortcuts Editor. The editor opens in a separate window.

**3** In the Data Shortcuts Editor window, click File menu ➤ Open Data Shortcuts Folder, and navigate to the project folder that contains the data shortcuts you want to edit.

The data shortcuts are displayed in a table grid.

# Using Vault

Autodesk Vault is a complete document management system for drawings, project objects, and project-related files.

Autodesk Vault provides access control and version control for all project drawings. One master copy of each project drawing is maintained in the project database. Other team members can reference the master copy, and they are notified when the master copy is updated.

Most project team members can access all required Vault features using the Prospector tree. However, you will need a knowledgeable system administrator who installs the server and client Vault software, uses ADMS Console or the Autodesk Vault administration tool to customize Vault for your purposes, and regularly performs system administration tasks such as data backups. Most of the topics in this section are geared toward your system administrator.

**NOTE** Some tasks, such as managing user accounts, can be done with either the Autodesk Vault tool or ADMS Console. When Help suggests using the Vault tool to perform a task, you can often use ADMS Console instead.

For information about installing Autodesk Vault, see the Autodesk Data Management Server Installation Guide.

Autodesk Vault provides access control and file version control. AutoCAD Civil provides commands in the Prospector tree to create and access project drawings and shared project data.

After you have logged in to Vault, each project is displayed as a collection on the Projects node of the Prospector tree..
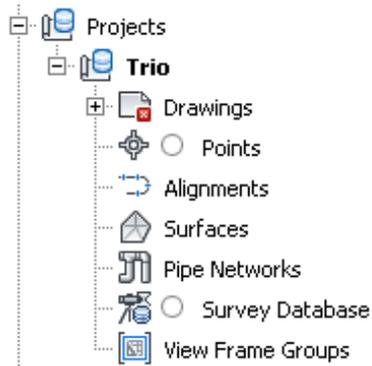


**Project Drawings**

A project contains a collection of project drawings. You can use folders to organize the drawings within a project. A master copy of a project drawing is maintained in a specified project within a Vault database, and only one person can edit the master copy at a time. When the edited project drawing is updated in the Vault database, a new version of the drawing is created. The icons displayed next to a drawing in the Prospector tree indicate the status of your local copy of the drawing in relation to the master copy of the drawing in the Vault database. For more information, see Project Item State Icons (page 140).

**Project Objects and References**

A project contains collections of project objects, including surfaces, alignments, profiles, pipe networks and view frame groups, that can be shared with others. All shared project objects are listed in the project object collections in the Prospector tree. Project members can create a read-only copy of a project object, called a *reference*, in a drawing. This read-only copy of the project object's geometry can be used to perform labeling, design, analysis, or what-if tasks.

Multiple project members can create references to the same shared project object.

**Project Points**

When using Vault, project management for AutoCAD Civil points is handled differently from project management for other objects. For more information, see Working with Vault Project Points (page 166).

**Project Management Commands**

The project management commands that are listed on Prospector shortcut menus can vary depending upon several factors, including the following:

■ Whether or not you have logged into Vault from the Prospector tree.

■ The state (for example, checked-in or checked-out) of the selected project item.

■ The location of the project item in the Prospector tree. For example, some drawing-related commands are available at the Open Drawings collection level, some are available at the Projects collection level, and some are available at both levels.

■ Whether or not the selected drawing has been saved.

■ Any circumstances that make the operation invalid for the selected item.

The user information that is defined for you on the current Vault server also affects the actions you are permitted to perform. For example, you might have permission to get copies of project drawings, but not create project drawings.

Vault Best Practices

# Project and Drawing Items in the Prospector Tree

The Master View of the Prospector tree contains three collections related to drawings and projects: Open Drawings, Projects, and Drawing Templates.
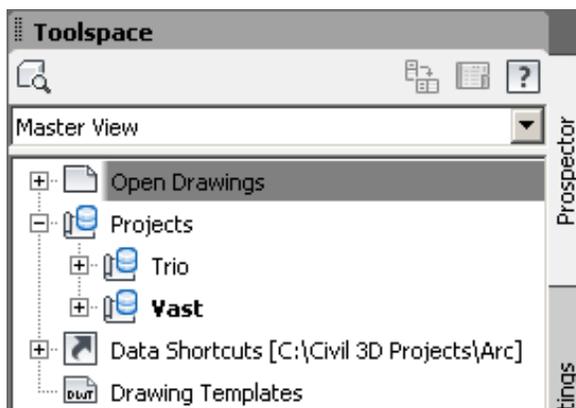
- **Open Drawings**. Lists open project and non-project drawings.

- **Projects**. Lists the projects available to you after you have logged in to a Vault server and a Vault database.

- **Drawing Templates**. Lists the available drawing templates. For more information, see Drawing Templates (page 109).

## Project Item State Icons

The project item state icons in the Prospector tree identify the state of a local project drawing in relation to the master copy in the Vault database.

NOTE To display the project item state icons, click  at the top of the Prospector tab.

The icons are displayed next to drawings that have been added to the project, indicating the state of the drawing on your system (your *local copy*) in relation to the latest version of the master copy of the drawing in the Vault database.

An icon that is displayed next to a project object (in a project object collection, such as Surfaces or Alignments) indicates the state of the project object's source drawing.

The project item state icons are shown in the following table:

| Icon | Meaning |
|---|---|
| | If no icon is displayed, then the drawing has been added to the project, but you do not have a local copy. |
| ○ | The drawing is available to be checked out, and the version of the master copy in the Vault database matches your local copy. |
| ● | The drawing is available to be checked out, but your local copy is newer than the latest version in the Vault database. This usually means that you changed the local copy without checking it out. |
| ● | The drawing is available to be checked out, and your local copy is older than the latest version in the Vault database. |
| ✔ | The drawing is checked out to you, but you do not have a local copy of the object. |
| ✔ | The drawing is checked out to you, and the version of the drawing in the Vault database matches your local copy. |
| ✔ | The drawing is checked out to you, and your local copy is newer than the latest version in the Vault database. This usually means that you have changed the drawing since you checked it out. |
| ✔ | The drawing is checked out to you, and your local copy is older than the latest version in the Vault database. |
| ✖ | The drawing is checked out to someone else, and you do not have a local copy. |

| Icon | Meaning |
|------|---------|
| ✖ | The drawing is checked out to someone else, and your local copy matches the latest version in the Vault database. |
| ✖ | The drawing is checked out to someone else, and your local copy is newer than the latest version in the Vault database. |
| ✖ | The drawing is checked out to someone else, and your local copy is older than the latest version in the Vault database. |

When your drawing contains a reference to a shared object, and the local copy of the source drawing becomes out of date, ⓘ is displayed next to the drawing name in the Prospector project tree. This generally occurs when someone adds a newer version of the project object's source drawing to the project.

## Drawing Item State Icons

The drawing item state icons in the Prospector and Settings trees identify whether an object is locked, is a reference, or is being referenced, as shown in the following table:

**NOTE** To display the drawing item state icons, click 🖾 at the top of the Prospector tab.

| Icon | Meaning |
|------|---------|
| 🖉 | Object is locked. |
| ◣ | Object is being referenced by another object. |
| ↗ | Object is a reference to an object in another drawing. |

## Drawing Item Modifier Icons

The drawing item modifier icons in the Prospector tree identify an object that is out of date or violates constraints, and also identifies the state of a local copy of a project object.

**NOTE** To display the drawing item modifier icons, click 🔰 at the top of the Prospector tab.

The following table explains the meaning of the three individual drawing item modifier icons:

| Icon | Meaning |
|------|---------|
| 🔰 | The object is out of date or violates constraints. |
| ◣ | The project object is more recent than the local copy. |
| ◢ | The local copy of the project object no longer matches the project version, which means that you have edited it. |

The following table lists all combinations of the drawing item modifier icons:

| Icon Displayed in Drawing | Object is Out of Date or Violates Constraints | Project Object is More Recent than Local Copy | Local Copy Has Been Edited Relative to Project Object |
|---|---|---|---|
| | No | No | No |
| ⚠ | Yes | No | No |
| ◣ | No | Yes | No |
| ⚠ | Yes | Yes | No |
| ◢ | No | No | Yes |
| ⚠ | Yes | No | Yes |
| ◡ | No | Yes | Yes |
| ⬟ | Yes | Yes | Yes |

**NOTE** Some of the above icons are displayed only in the Points list view.

## The Project Collection List Views

Use the list views displayed in the Projects collection to obtain information about the drawings and shared objects in the project.

**NOTE** To view the Projects collection in the Prospector tree, select Master View from the list at the top of the Prospector tab.

Click a collection in the Prospector tree Projects collection to display a list view.

If the selected item is a project, the following information is displayed in the list view:

- Project name
- Description
- User name of the person who created the project
- Date the project was created

If the selected item is a collection or folder containing drawing files, the following information is displayed in the list view:

- Name
- Version

- User name of the person who created this version

- Date checked in

- Comment

If the selected item is a collection containing shared objects, the following information is displayed in the list view:

- Object name

- Source drawing name

- Description

For more information about changing the list view display, see Customizing a List View (page 84).

# Installing and Configuring Vault

The Autodesk Data Management Server Installation Guide contains detailed recommendations for configuring your Vault installation.

A single-site configuration is recommended for most teams. The Autodesk Data Management Server (ADMS) is installed on one system and the client component is installed on each computer that must communicate with that server. The server stores all master copies of data files, and the clients can access the files stored on the server. For best performance, a dedicated server is recommended.
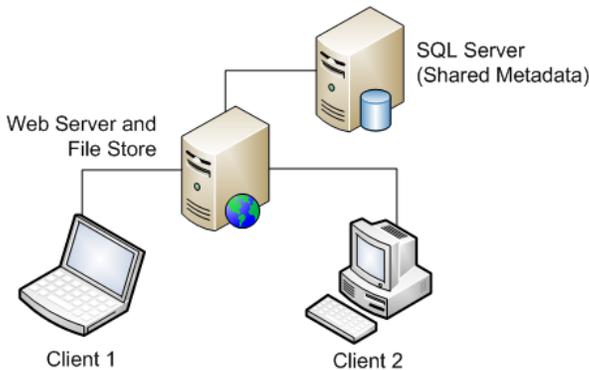


**Vault single-site architecture**

You have flexibility within this model to choose a configuration that suits your requirements, based on criteria such as the number of users who must share data, how your users are divided into design teams, and the amount of project data you expect your users to generate. Refer to the Autodesk Data Management Server Installation Guide for more details.

After Vault is installed, you can decide to create a single vault (database) or multiple vaults on the server. User permissions apply within a single vault, and cannot be shared across vaults. For more information on the difference between a single vault configuration and a multiple vault configuration, see Autodesk Data Management Server Installation Guide.

Vault's optional multi-site feature is designed to support geographically dispersed workgroups sharing vaults. A single instance of remote SQL manages multiple servers, each with its own file store, at separate office sites. Depending on their level of permissions, all users can see all files at all sites as if the system was a single database.
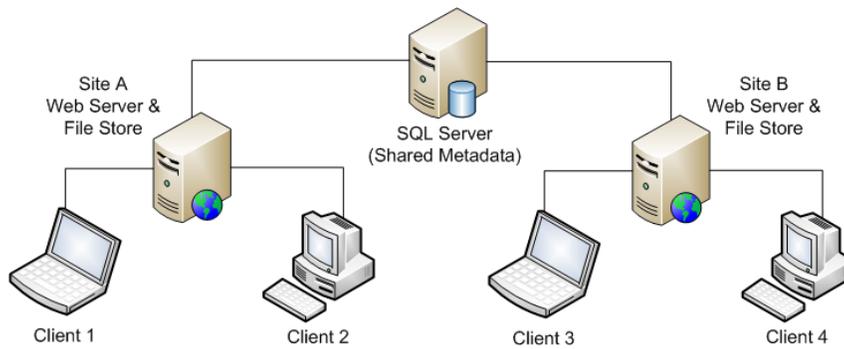
**Vault multi-site architecture**

When a user checks out a file that is stored at another site, the system replicates that file at the user's site. As long as the file remains shared, the system remembers to update that copy with any changes. Vaults and individual files are replicated only where they are really in use. Backup and restore operations can be managed from any site that has all vaults enabled. For more information, see ADMS Console Help.

# Project Folders

Any file (drawing or not) that is associated with a project, especially any file that would benefit from version control, should be stored in the project. Temporary files, such as preliminary design drawings, do not need to be stored in the project.

For ease of use, all projects should use consistent folder structure and folder names. A practical structure is provided when you use the project template. You can easily create your own custom template if the default version is not suitable.

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**NOTE** In general, a user who creates a folder or project data can delete it using the Autodesk Vault tool. For this reason, it is recommended that all new projects be created by a system administrator. This will prevent other users from advertently deleting a project and associated files.

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You do not need to create a folder for point database files or survey data. These are automatically stored directly beneath the main project folder.

A project can also include files that are not drawings. Using the Autodesk Vault tool, create folders for such files directly under the project folder. Possible folder names might include: Proposals, Client Correspondence, and Contracts. You can store and access the files in non-drawing folders using the Vault tool.

**Working Folders**

When you work on objects, you check associated drawings out of a vault and work with them from working folders. The objects in a working folder are physically copied from a Vault file store. Other users can still get read-only copies of the files, but they cannot edit them until you have checked them back in. In this way, you maintain a single master version of each drawing.

When you check a file back into a vault, the revised version from your working folder is copied to the file store on the server. The Vault database marks the file with the appropriate revision data and makes it available to other users.

You can change your working folder without risk to the data in the Vault database, but you typically change the working folder only if you need to work with projects in different Vault databases. To avoid confusion, use a separate working folder for each Vault database.

The default working folder location is C:\Civil Projects.

For more information on best practices with working folders, see Working Folder Configurations in the *AutoCAD Civil Best Practices* guide.

# Controlling Access to Project Data

You can control who can access and edit project drawings and master copies of project objects.

To ensure a secure environment, define a separate Vault user account for each person who will be accessing project data on the project server. Each account requires a unique user name and password. For more information, see Creating Vault User Accounts (page 147).

Vault allows you to control access to project files at several levels: user names, groups, projects and folders. With careful planning, you can minimize the amount of configuration required and create a flexible security system. It is recommended that you control access to project files at the group level.

Create a different Vault group for each function performed by members of your organization. For example, you could create a set of groups including: Surveyors, whose members would create and edit master copies of data in a folder such as EG Surfaces; Civil Engineers, whose members would usually read (but not modify) data in the EG Surfaces folder, and would create and edit master data in a folder such as Alignments; and Drafters, whose members would generally read data but would not modify master copies of data. Access can then be further restricted to individual groups or users, at the folder level.

In general, each team member should belong to only one group. To change how someone accesses project files, move their user name from one group to another. For more information see Creating Vault Groups (page 147).

# Partitioning Project Data

You should divide your project data across multiple drawings using a strategy that both protects data integrity and supports ease of access.

In AutoCAD Civil, there is one master copy of each project object (such as a surface, alignment, profile, or pipe network), and that master copy is stored in a source drawing that is stored in the project's Vault database. There are virtually no restrictions on how many project objects you can store in the same drawing, but there are some factors you should consider.

A key determining factor in how you partition your project data is the ability of team members to create and modify project data when they need to. Because only one person can edit a drawing at a time, placing the master copy of every project object into the same drawing is too restrictive for a collaborative environment. In general, spreading your project objects across many drawings makes more sense than crowding many objects into fewer drawings.

For each project object, create a separate drawing that contains the geometry of the project object and, if needed, related project objects. Consider placing each surface in a separate drawing. At the very least, consider placing a key surface, such as an existing ground surface, in its own drawing. You should avoid placing all your project alignments in the same drawing.

The function of a drawing should match its content. If a drawing contains the master copy of a surface object, that drawing shouldn't also be a plan profile sheet. Keep project object geometry in its own drawing and create separate drawings for viewing and annotating read-only copies of the geometry.

It is important to understand that a single project must contain all the data that is required for that project. A drawing cannot contain references to more than one project at a time.

Adhering to a consistent naming convention will make the drawings that contain master copies of project objects easy to identify. Name a drawing that contains a project surface with the same name as the surface; name a drawing that contains a project alignment with the name of the alignment. Store project drawings in appropriately named folders. For more information see Project Folders (page 144).

Develop procedures that minimize the number of shared project objects. Make sure that team members understand that when adding or checking in a drawing, they should not share an object unless it is needed by others.

# Running the Autodesk Vault Tool from the Prospector Tree

You can access the Autodesk Vault tool from the Prospector tree.

You must log in to Vault from the Prospector tree before you can run the Vault tool from the Prospector tree.

**To run the Vault tool from the Prospector tree**

1  In Toolspace, on the Prospector tab, right-click the Projects collection.

2  Click Autodesk Vault.

3  If the Welcome dialog box is displayed, click Log In.

4  In the Log In dialog box, enter your Vault user name.

5  Enter your password.

6  Select a Vault server.

7  Select a Vault database. Click ⬚ to list available databases.

8  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): right-click Projects collection ➤ Autodesk Vault

# Running Vault from Microsoft Office Applications

The Microsoft Office add-in for Autodesk Vault enables direct access to Vault from Word, Excel or PowerPoint documents.

However, the Office add-in is not installed by default when you install Vault.

**To install the Office add-in after Vault is installed**

1  In Windows, from the Start menu, open the Control Panel. Depending on your system, the Control Panel may be on a Settings submenu.

2  Click Add or Remove Programs, or in Vista, click Uninstall a Program.

3  In the list of programs, click Autodesk Vault.

4  Click Change/Remove, or Uninstall/Change.

5  In the Autodesk Vault Installation Wizard dialog box, click Add or Remove Features.

6  In the Select Features to Install field, select the MS Office Add-In. Click Next.

7  Click Next.

8  Click Finish.

To use Vault with a Microsoft Office application, click File menu ➤ Autodesk Vault and select one of the Vault options.

# Project Files Created in a Project Database

When an AutoCAD Civil project is created, an initial set of empty or near-empty files is created in the project database.

Data is added to these files as you add drawings and data to the project.

### Project.xml

This file contains information about the project and the shared project objects. The presence of this file is what indicates to AutoCAD Civil that the folder containing this file is an AutoCAD Civil project folder. Do not delete, modify, or move this file.

### PointsGeometry.mdb

This file contains the project's point geometry. The points contained in the file are listed in the project's Points list in the Prospector tree. This file is not checked in and out directly as project drawing files are; it is checked in and out indirectly when users use point access control commands.

### PointsStatus.mdb

This file contains access control information about the points in *PointsGeometry.mdb*, including the status (checked-in, checked-out, or protected) of individual points.

### Survey.sdb

This file represents the project survey database. For more information, see Survey Database (page 196).

# Creating Vault User Accounts

Each team member must have a user account on the project Vault server before using project management commands.

To ensure a secure environment, define a separate Vault user account for each person who will be accessing project data on the project server. Each account requires a unique user name and password.

**NOTE** Each instance of a Vault Data Management server has a separate set of user accounts. If a user requires access to multiple servers, you must define an account for him on each server.

Only a user with administrator privileges can add a user. When you add a new user to a server, you can specify a *role* for the user, which defines how the user will access data on the server. AutoCAD Civil Vault project management supports *administrator, editor,* and *consumer* roles for user accounts.

In general, you will find managing access to project data easier to control at the group level, instead of at the individual user level. When you create a user account, specify a role for each user that gives them the highest level of access they require to any part of the database. For more information, see Creating Vault Groups (page 147).

**To add a new user in the Autodesk Vault administration tool**

■ Click Tools ➤ Administration and then use the Security tab to add a new user.

For more information, see Add/Edit Users in Autodesk Vault help.

# Creating Vault Groups

Use Vault groups to manage and control access to project files.

Use the Autodesk Vault tool to add a group to a Vault server. Only a user with administrator privileges can add a group.

When you create a group, you specify generally how the group's members should access project data by assigning the group the role of *data editor*, which means that members of the group can add or edit data, or *data consumer*, which means the group members can only read data, not edit it.

In general, a user should belong to only one group. To change someone's access to project files, move their user name from one group to another.

To add a group in the Vault tool, click Tools menu ➤ Administration and then use the Security tab to add the group. You can also add users to a group here. For more information, see Managing Groups in Autodesk Vault help.

# Folder Permissions

With Autodesk Vault in AutoCAD Civil, system administrators can extend permissions down to folders and sub-folders in a vault.

Before setting folder permissions, you may want to manage specific Vault users, groups, or roles from the Tools menu in the Autodesk Vault tool.

**To set folder permissions**

1 In the Vault tool window, expand the Vault Explorer ($) tree to the folder whose permissions you want to set.

2 Right-click the folder. Click Properties.

3 To add a user or group to the Access Control List, in the Properties dialog box, Security tab, click Add.

4 In the Add Members dialog box, select the member(s) to add to the Access Control List. Click Add.

5 Click OK.

6 In the Properties dialog box, click OK.

# Pack and Go

Use the Vault Pack and Go feature to hand off project data to someone else.

Pack and Go resolves file dependencies for a drawing. This feature can be used to supply copies of project files to someone. It is not intended for making copies of project files that you want to edit and then return to the project.

For example, if your firm begins a project by doing a survey, creating the base map, and completing the parcel layout, and then passes the drawing and associated data to a civil engineer who completes the job, you could use Pack and Go in Autodesk Vault to assemble the information related to the drawing before you hand it off.

For more information, see Packaging Files in Autodesk Vault help.

# Working with Vault Projects

An AutoCAD Civil project groups all the drawings and data for a single project in one place and controls access to all project files.

After you log in to a Vault server and a Vault database, the Projects collection in the Prospector tree lists all the projects that are available to you.

**NOTE** You must select Master View from the list at the top of the Prospector tab before you can view the Projects collection.

## Logging in to a Vault Server

Before you can create or access project data, you must log in to a Vault server and a Vault database from the Prospector tree.

You must be an authorized user on the server before you can log in. The server and the database that you select when you log in determines what projects you can access.

Before you log in, project management commands are not available on shortcut menus in the Prospector tree. After you log in, the projects available in the specified server and Vault database are listed in the Prospector tree, and applicable project management commands are displayed on the menus when you right-click a project item in the Prospector tree.

**To log in to Vault from Prospector**

1 In Toolspace, on the Prospector tab, select Master View from the list at the top of the Prospector tab.

2 Right-click the Projects collection.

3 Click Log In.

4 In the Log In (page 2205) dialog box, enter your Vault user name.

5 Enter your password.

6 Select a Vault server.

7 Select a Vault database. Click ⌶ to list available databases.

8 Optionally, select Automatically Log In Next Session.

9 Click OK.

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab (Master View): right-click Projects collection ➤ Log In

Dialog Box

  Log In Dialog Box (page 2205)

## Logging Out of Vault

Logging out of Vault terminates your connection with the Vault server and the Vault database that you are logged in to.

After you log out, you can no longer view or access project data in Prospector. When you end an AutoCAD Civil session, you are automatically logged out.

**To log out of vault**

1 In Toolspace, on the Prospector tab, right-click the Projects collection.

2 Click Log Out.

**Quick Reference**

## Changing the Working Folder

The working folder maps your local copies of project files to the master copies of project files in the Vault database.

The default working folder for Autodesk Vault projects is C:\Civil Projects. This same default location is also used for data shortcut projects. For ease of management, it is a best practice in most cases to use different working folders for the two project types.

The working folder contains your local copies of files for all the projects stored in a Vault database. If all your projects are stored in the same Vault database, you will use the same working folder for all projects. If you work with projects that are stored in different Vault databases, you can prevent confusion by using a different working folder for each Vault database.

In general, you should not change your working folder while you have any objects referenced or files checked out from a Vault project. The working folder is used to determine the state of your local copies of files in relation to the master copies in the project database.

**To change the working folder**

1   In Toolspace, on the Prospector tab, right-click the Projects collection.

2   Click Properties.

3   In the Properties - AutoCAD Civil Projects (page 2207) dialog box, under Working Folder, click ⊡ .

4   In the Browse For Folder dialog box, browse to the folder and click OK.

## Quick Reference

## Creating Projects

If you are logged in to Vault, you can use the Prospector tree to create a new project in the project database.

You create the project using either the default Sample Project template, or another template you have saved. Project templates provide the capability to create a new project with a predetermined folder structure, where each folder can also contain files of any type. The template folders are found in the AutoCAD Civil Project Templates folder on your local disk.

**NOTE** In general, a user who creates a folder or project data can delete it using the Autodesk Vault tool. For this reason, it is recommended that all new projects be created by a system administrator. This will prevent other users from inadvertently deleting a project and associated files.

**To create a project**

1  In Toolspace, on the Prospector tab, right-click the Projects collection.

2  Click New.

3  In the New Project (page 2206) dialog box, enter a project name.

4  If you want to create your new project from a template, select Use Project Template.

5  On the Project Template list, click the template you want to use.

6  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): right-click Projects collection ➤ New

Dialog Box

New Project Dialog Box (page 2206)

# Project Administration

Your system administrator can use ADMS Console or the Autodesk Vault tool to customize Vault for your purposes and to regularly perform project administration tasks on an on-going basis.

These tasks include the following:

■  Creating, modifying, or deleting folders in a project

■  Deleting a project

■  Modifying the state of a file, for example, checking in a file that for some reason cannot be checked in by the person who originally checked it out

■  Restoring a file back to a prior version

■  Backing up project files

For more information, see the ADMS Console Help and Administrative Tasks in Autodesk Vault help.

# Changing the Active Project

The active project is displayed in bold type in the Prospector tree. It is the default project for the project dialog boxes.

**NOTE**  The active project is not displayed in the Select Projects To Display (page 2207) dialog box.

**To change the active project**

1  In Toolspace, on the Prospector tab, expand the Projects collection and right-click a project.

2  Click Set Active.

### Quick Reference

Prospector tab (Master View): Projects collection ➤ right-click &lt;project name&gt; ➤ Set Active

## Selecting Projects to Display

You can limit the number of projects that are listed in the Prospector tree.

The projects available to you are determined by the server and the database that you are logged in to. By default, all projects available in the database are listed in the Prospector tree under the Projects collection.

Reducing the number of projects that are listed in the Prospector tree will not only reduce the amount of data you will see, but also can reduce the time required to update the display of the tree.

You cannot turn off the display of the active project, and the name of the active project is not listed in the dialog box.

**To select projects to display**

1   In Toolspace, on the Prospector tab, right-click the Projects collection.

2   Click Select Projects To Display.

3   In the Select Projects To Display (page 2207) dialog box, select the check box next to the name of any project that you want listed in the Prospector Projects collection.

> **NOTE**  The name of the active project is not included in the list.

4   Click ![icon] to select all the projects in the list.

5   Click ![icon] to clear all the projects in the list.

6   Click OK.

### Quick Reference

Prospector tab (Master View): right-click Projects collection ➤ Select Projects To Display

Dialog Box

Select Projects To Display Dialog Box (page 2207)

## Working When Disconnected

You can work with project files while you are disconnected from Vault.

If you are not connected to the network, in general, you can work with your local copies of project drawings the way you normally do. However, you cannot perform project management functions (such as checking in and checking out files), you cannot get updated versions of any project files, and you cannot create new references to project objects while you are disconnected.

Before you disconnect, you should update your local copies of any project files by getting the latest versions of any project files that you are working with, including the source drawings for any project objects that your drawing references.

## Deleting a Project

Delete a project from the project database using the Autodesk Vault tool.

A project can be deleted by a system administrator or by the user that created it. It is recommended that all projects be created by a system administrator to protect projects from being inadvertently deleted by users who are not system administrators.

To delete a project from the project database, delete the top-level project folder. You cannot delete a file that is checked out.

After a project has been deleted from the Vault database, its name remains in the Projects collection in the Prospector tree until you refresh the display of the tree.

After the project has been deleted from the Vault database, use Windows Explorer to delete local copies of project files and folders from the working folder.

## Converting a Project to Data Shortcuts

Change all data references from the Vault format to data shortcuts.

After getting the latest version of all files from the Vault project, use the Data Shortcut Editor to convert the Vault project.xml file into a _Shortcuts folder. Doing this converts the Vault references into data shortcuts between drawings.

**To convert a Vault project**

1 On the Windows Start menu, click Programs ➤ Autodesk ➤ AutoCAD Civil 2010 ➤ Data Shortcuts Editor.

2 In the Data Shortcuts Editor, click File menu ➤ Open Data Shortcuts Folder. In the Browse for Folder dialog box, browse to the folder for the project that you want to convert. The default location is C:\Civil Projects.

   After you select the project folder, the Vault references are loaded into the table.

3 In the Data Shortcuts Editor, click Edit menu ➤ Find and Replace. Replace each instance of $ with the path to the local project folder (example: C:\Civil Projects).

4 In the Data Shortcuts Editor, click File menu ➤ Save As, and then save the new Shortcuts folder in its current location.

5 In the active project folder, delete the project.xml file.

## Labeling Project Milestones

You can use the Vault labeling feature to identify the status of a set of project files, such as the project milestone attained.

When a major project milestone is reached, you can label the files involved so they can be easily identified. When you create a label, the most recently checked-in version of every file in the project is assigned that label.

Labeling does more than just tag a set of files. If you later need to archive or retrieve the set of files, you can do so. Labels do not include folder information; only the project files are labeled.

To create a label, in the Autodesk Vault tool, right-click a project folder and click New Label. For more information on creating and viewing labels, see in the Autodesk Vault tool help.

## Exporting a Project

Export a project to a ZIP file for storage, or to move the project to another vault.

When a project is exported, copies of the latest versions of all drawings and other documents are compressed and saved in a ZIP archive. This archive can be reliably imported to a vault, retaining the data references between drawings.

The export operation does not affect the original project documents. They remain in place for ongoing use.

**TIP** Label your project files, then export them to archive a copy of the files at a specific milestone.

**See also:**

■ Importing a Project (page 154)

■ Labeling Project Milestones (page 153)

**To export a set of project files**

1 In Toolspace, on the Prospector tab, in Master View, expand the Projects node to see the list of projects.

2 Right-click the project to export and click Export to ZIP File.

3 In the Browse for Folder dialog box, navigate to and select the folder where you want to save the ZIP file, then click OK.

The system displays a progress bar while the files are compressed and saved.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View) ➤ right-click Project ➤ Export to ZIP File

## Importing a Project

You can import a Vault project or a project that uses data shortcuts.

Data shortcuts are converted to Vault references during import.

The project to be imported can be in a folder or a ZIP file. If you are importing a project that was previously exported from a vault, it is in a ZIP file.

You cannot import a project if it has the same name as an existing project in the vault.

**See also:**

■ Exporting a Project (page 154)

**To import a project**

1 In Toolspace, on the Prospector tab, in Master View, right-click the Projects node ➤ Import From, and click either ZIP File or Folder.

2 In the dialog box that opens (Import Project Archive or Browse for Folder, depending on project format), navigate to and select the project container, and then click OK.

The system displays a progress bar and status messages during the import operation.

### Quick Reference

## Managing File Status Prompts

You can control the use of prompts during file management events.

During file operations such as Check In, and Get Latest Version, prompts are available to advise you of unsaved changes, newer versions in the vault, and other details. You can specify whether these prompts are used always or never. You can also specify whether the default response to the prompt is Yes or No.

A new user can have all prompts displayed to guide them in file management. An experienced user can suppress some unnecessary prompts and work more efficiently.

**To manage file status prompts**

1   In Toolspace, on the Prospector tab, in Master View, right-click the Projects node ➤ Manage Prompts.

2   In the Manage Prompts dialog box, review the Response and Frequency settings for each prompt, making any required changes.

### Quick Reference

## Migrating Old Civil 3D Projects

You can easily migrate projects to AutoCAD Civil 2010 from AutoCAD Civil 3D 2006 and previous versions.

You must log in to Vault from the Prospector tree before the project migration command is available.

**To migrate an old Civil 3D project**

1   In Toolspace, on the Prospector tab, right-click the Projects collection.

2   Click Migrate Pre-Civil 3D 2007 Project.

3   In the Select Civil 3D Project Folder To Migrate dialog box, browse to the project folder to be migrated.

4   Select the folder that contains the project and click Open.

The new project is created on the Vault server in the database that you are logged in to.

### Quick Reference

# Working with Vault Project Drawings

Use drawings to create and access project data.

There is a tight association between a project object, such as a surface, and its *source drawing*, which is the drawing that contains the object. To create a project object, add the object's source drawing to a project and specify that you want to share the object with others. You can edit a project object by checking out the source drawing, making the change, and checking the source drawing back in.

You can store any project drawing in the project database. You will find it easier to keep your project organized if you use more than one folder to store your project drawings. For more information, see Project Folders (page 144).

The Drawings collection in the Prospector tree is not a folder and does not map to a folder in either the project database or the working folder. However, the folders under the Drawings collection map directly to folders in the project database and the working folder.

## Adding a Drawing to a Project

You can add a drawing to a project to make the drawing available to others, to place it under version control, and to create project objects from objects in the drawing.

**TIP** Before adding a drawing to a project, save the drawing in the most appropriate folder in the project working folder. Doing this ensures that Vault can determine where to save the drawing in the database.

While adding a drawing to a project, specify which objects (if any) in the drawing you want to share with others. Those objects are then added to the appropriate Prospector project object collection (such as Surfaces, Alignments, or Pipe Networks) and can be referenced by others.

**NOTE** You should only share objects that are required by others. If you are unsure whether an object should be shared, do not share it. You can share it later using the Check Out and Check In commands.

While adding a drawing to a project, you can specify that associated files, including surface data files, image files, and xrefs, are added to the project. You can also specify whether a *.dwf* file is created and added to the project for display in drawing item views and the Autodesk Vault tool.

If a drawing containing a surface is added to a project, and the surface references external data files, a folder is created under the drawing to hold those files. The files are placed in the folder, and the surface definition in the drawing is updated to reflect the new (relative) location of the files.

An open drawing must be saved before you can add it to a project. A copy of the drawing is created in your local working folder and the master copy is added to the project.

**To add a drawing to a project**

1   In Toolspace, on the Prospector tab, expand the Open Drawings collection.

2   Right-click the drawing you want to add to the project.

3   Click Add to Project.

4   On the Select A Project (page 2196) page of the Add To Project wizard, select the project to which you want to add the drawing.

5   Click Next.

6   On the Select A Drawing Location (page 2196) page of the Add To Project wizard, specify the location of the drawing within the project. Click [icon] to create a new folder.

7   Click Next.

**8** On the Drawing File Dependencies (page 2196) page of the Add To Project wizard, specify file dependencies and *.dwf* creation options.

**9** Click Next.

**10** In the Share Data (page 2196) page of the Add To Project wizard, specify which objects in the drawing you want to share with others.

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**NOTE** If the drawing does not contain objects that can be shared, this page is not displayed.

---

**11** Click Finish.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Open Drawings collection ➤ right-click <drawing name> ➤ Add to Project

Add To Project Wizard

## Checking a Drawing out from a Project

You can check out a project drawing to edit it.

Checking out a drawing prevents others from changing the drawing while you are working on it. When you have completed your changes, check in the drawing.

Check out a drawing only if you intend to change it. If you do not need to change the drawing, get a read-only copy using Get Latest Version. For more information, see Getting the Latest Version of a Drawing (page 159).

You can check out files that the drawing is dependent on when you check out the drawing. However, you should only check out dependent files if you intend to change them.

When you check out a drawing, a local copy of the drawing is placed in your working folder, and the drawing is opened.

**To check a drawing out from a project**

**1** In Toolspace, on the Prospector tab, under the Projects collection, expand the project collection and the project Drawings collection that contains the drawing you want to check out.

---

**NOTE** If a drawing is open under the Open Drawings collection, all right-click functions are transferred to that location.

---

**2** Right-click the drawing and click Check Out.

**3** In the Check Out Drawing (page 2198) dialog box, specify whether dependent files should also be checked out.

**4** Specify whether you want the latest version of the file.

Getting the latest version overwrites your local copy of the file with the most recent version of the master copy of the file. Most of the time, you should get the latest version of the file. However, if you want to create a new version of the file from your existing local copy, do not get the latest version.

**5** If Include File Dependencies is selected, specify which project files should be checked out.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Drawings collection ➤ right-click <drawing name> ➤ Check Out

Dialog Box

Check Out Drawing Dialog Box (page 2198)

## Checking a Drawing in to a Project

After you check out a project drawing and change it, check in the drawing to make it available to others and to update the version in the project database.

You can only check in a drawing that you have checked out. Checking in the drawing returns the file to its original project database and folder, and updates the version. If the drawing is opened and has been changed, it is automatically saved before it is checked in.

When checking in a project drawing, you can specify which objects in the drawing (if any) you want to share with others. You can also specify whether a *.dwf* file is created for display in drawing item views and the Autodesk Vault tool.

**To check a drawing in to a project**

**1** In Toolspace, on the Prospector tab, under the Projects collection, expand the project collection and the project Drawings collection that contains the drawing you want to check in.

**NOTE** If a drawing is open under the Open Drawings collection, all right-click functions are transferred to that location.

**2** Right-click the drawing and click Check In.

**3** On the Drawing File Dependencies (page 2197) page of the Check In Drawing wizard, specify any file dependencies, *.dwf* creations options, and comments.

**4** Click Next.

**5** On the Share Data (page 2197) page of the Check In Drawing wizard, specify which objects in the drawing you want to share with others.

**NOTE** If the drawing does not contain objects that can be shared, this dialog box is not displayed. Objects that were shared during the Add to Project command or a previous Check In command are not listed.

**6** Click Finish.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master view): Projects collection ➤ project Drawings collection ➤ right-click <drawing name> ➤ Check In

OR

Prospector tab: Open Drawings collection ➤ right-click <drawing name> ➤ Check In

Check In Drawing Wizard

## Opening a Project Drawing

After a drawing has been added to a project, open it from the Projects collection in the Prospector tree.

If you want to open a project drawing and edit it, you must first check it out. If you do not have the project drawing checked out, you can only open it in read-only mode.

You can tell whether the drawing is checked out to you by the icon displayed next to the drawing name in the Prospector tree. For more information, see Project Item State Icons (page 140).

If the drawing you want to check out is a source drawing for a project object, you can open it from the project object collection. For more information, see Viewing Project Object Source Drawings (page 164).

**To open a project drawing**

1   In Toolspace, on the Prospector tab, under the Projects collection, navigate to the folder that contains the drawing you want to open.

2   Right-click the drawing.

3   Click Check Out or Open (read-only).

The command that is listed on the menu depends on whether or not you have the drawing checked out.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Drawings collection ➤ right-click <drawing name> ➤ Open

## Getting the Latest Version of a Drawing

You can obtain a read-only copy of the latest version of any project drawing.

Use the Get Latest Version command to update your copy of a source drawing for a project object, a drawing that contains a reference to shared project objects, or any other project drawing.

You can tell whether your local copy of a drawing is the latest version available in Vault by the icon displayed next to it in the Prospector tree. For more information, see Project Item State Icons (page 140).

The Get Latest Version command is displayed in the shortcut menu of a Prospector tree item only when the local copy of the file is out of date or does not exist. The command is not available for an open drawing.

When applicable, the Get Latest Version command is available at every level within the Projects collection on the Prospector tab. If you select the command at a level that contains folders beneath it, you can choose which folders you want to update.

**To get the latest version of a drawing**

**1** In Toolspace, on the Prospector tab, under the Projects collection, right-click the drawing, collection, or folder you want to update.

**2** Click Get Latest Version.

**3** In the Get Latest Version (page 2204) dialog box, select Include File Dependencies to display any dependent files in the Project Files list.

**4** Clear the check boxes next to files and folders that you do not want updated.

**5** Click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ right-click collection, folder, or item ➤ Get Latest Version

Dialog Box

Get Latest Version Dialog Box (page 2204)

## Resetting a Checked out Drawing

If you have a project drawing checked out, you can reset it to a checked-in state without making any changes to it.

A drawing must be checked out to you before you can reset its state. The Undo Check Out command resets your local copy of the drawing back to the latest version of the file in the project database, which means that any changes you made to the drawing are lost. You can preserve the changes you made to the drawing by saving and renaming the drawing (using Save As) before using the Undo Check Out command.

The status of the drawing in the project database is reset to checked-in, and the file version is not updated.

**To undo the checkout of a drawing**

**1** In Toolspace, on the Prospector tab, under the Projects collection, expand the project collection and the project Drawings collection that contains the drawing you want to reset.

> **NOTE** If a drawing is open under the Open Drawings collection, all right-click functions are transferred to that location.

**2** Right-click the checked-out drawing.

**3** Click Undo Check Out.

**4** In the Undo Check Out (page 2209) dialog box, select Include File Dependencies to display dependent files.

**5** If dependent files are displayed, clear the check boxes next to files that should remain checked-out.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Drawings collection ➤ right-click <drawing name> ➤ Undo Check Out
OR
Prospector tab: Open Drawings collection ➤ right-click <drawing name> ➤ Undo Check Out

Dialog Box

Undo Check Out Dialog Box (page 2209)

## Synchronizing Drawing Data with Project Data

You can update an open drawing with the latest versions of all project objects and point data.

The Sync To Project command updates out-of-date drawings that reference project objects. It also updates any out-of-date project points contained in the drawing. The out-of-date condition can occur when someone checks in a project object's source drawing or checks in project points.

Local copies of out-of-date source drawings and project points are replaced with the latest version from the project database.

**To synchronize a drawing**

1  In Toolspace, on the Prospector tab, under the Open Drawings collection, right-click the drawing you want to synchronize.

2  Click Sync To Project.

3  In the Sync To Project (page 2208) dialog box, select the out-of-date project objects you want synchronized.

4  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Open Drawings collection ➤ right-click <drawing name> ➤ Sync to Project

Dialog Box

Sync To Project Dialog Box (page 2208)

## Detaching a Drawing from a Project

You can detach a drawing from a project to sever the connection between the drawing and all project data.

**WARNING** When you detach a drawing from a project, all links to project source drawings and project data are severed and cannot be re-established.

A drawing must be open to be detached from a project. When you detach a drawing from a project, all references in the drawing to shared project objects are replaced with object data that can be edited.

When a drawing contains a project object, such as a reference to a project surface, the project object is promoted to a surface object that has no reference back to the original source drawing that its geometry it based on. You can edit the promoted surface, but you cannot rebuild it based on the original source data.

**To detach a drawing from a project**

1   In Toolspace, on the Prospector tab, in the Open Drawings collection, right-click the drawing you want to detach.

2   Click Detach Project.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Open Drawings collection ➤ right-click <drawing name> ➤ Detach Project

# Working with Vault Project Objects

You can create, manage, and reference project objects through their source drawings.

A project object is an object that is intended to be shared with others working on a project. A project object is managed through its *source drawing*, which is the drawing that contains the object.

A project object becomes available to others when the source drawing containing the object is added to or checked in to the project database and the object is selected to be shared.

To include a project object in another drawing, you create a *reference* to the object. A reference is a read-only copy of the object that you can refer to as if it were the object itself.

## Creating Project Objects

Create a project object by adding a source drawing to a project.

When you add or check in a drawing to a project, you can specify which objects (such as surfaces) in the drawing are to be shared. For more information, see Adding a Drawing to a Project (page 156) and Checking a Drawing in to a Project (page 158).

After a project object has been shared, it is listed in the project's object collection (such as Surfaces, Alignments, or Pipe Networks) in the Prospector tree.

You can access a project object's data in another drawing by creating a read-only copy called a reference. For more information, see Creating References to Project Objects (page 162).

## Creating References to Project Objects

Include a read-only copy of a project object in a drawing by creating a reference to it.

All the available project objects in a project are listed in the project's object collections, including Surfaces, Alignments, and Pipe Networks, in the Prospector tree.

You access a project object's data in a drawing by creating a reference to it. A reference is a read-only copy of an object. When you create a reference to an object, you cannot modify the object's geometry, but you can access its properties and data, apply styles and annotation, and perform some types of analysis.

Creating a reference creates a local copy of the object's source drawing in your working folder. If you already have a local copy of the object's source drawing, it is updated to the latest version, if necessary.

**To create a reference to a project object**

1   Open the drawing to which you want to add the reference, and make it the active drawing.

2   In Toolspace, on the Prospector tab, expand the Projects collection.

**3** Expand the project and object collections that contain the project object.

**4** Right-click the project object.

**5** Click Create Reference.

**6** In the Create Reference dialog box for the object, specify how the object's geometry will be displayed in the drawing.

**7** Click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Create Reference

# Removing a Project Object

Make an object unavailable for referencing in other project drawings.

You can delete an object from the list of reference objects for a project. Any existing references to the object remain intact, and continue to be updated from the source object.

**To delete a project object**

**1** In Toolspace, on the Prospector tab, expand the Projects collection.

**2** Expand the project and object collections that contain the project object.

**3** Right-click the project object to be deleted.

**4** Click Remove.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Remove

# Getting the Latest Version of Project Objects

If someone makes a change to a project object, you can easily update your local copy of the object's source drawing.

When someone creates a new version of a project object's source drawing in the project database, your local copy becomes out of date. You can determine the state of your local copy by the icon displayed next to either the source drawing or the project object in the Prospector tree. For more information, see Project Item State Icons (page 140).

**NOTE** The Get Latest Version command is available at all levels of a project in the Prospector tree, when applicable.

**To get the latest version of a project object**

**1** In Toolspace, on the Prospector tab, expand the Projects collection.

**2** Expand the project collection that contains the project object.

**3** Expand the object collection that contains the project object.

**4** Right-click the project object and click Get Latest Version.

**5** In the Get Latest Version (page 2204) dialog box, select Include File Dependencies to display dependent files in the Project Files list.

**6** Clear the check boxes next to files and folders that you do not want updated.

**7** Click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Get Latest Version

Dialog Box

Get Latest Version Dialog Box (page 2204)

# Viewing Project Object Source Drawings

You can quickly open or switch to a project object's source drawing.

You can open a project object's source drawing from the project object level in the Prospector tree. If the drawing is already open, you can make it the active drawing.

**NOTE** If you do not have the drawing checked out, you can only open the drawing in read-only mode.

**To access a source drawing for a project object**

**1** In Toolspace, on the Prospector tab, expand the Projects collection.

**2** Expand the project collection that contains the project object.

**3** Expand the object collection that contains the project object.

**4** Right-click the project object and do one of the following:

■ If you have the source drawing checked out, click Open Source Drawing to open it.

■ If you do not have the source drawing checked out, click Open Source Drawing (Read-only) to open it.

■ If the drawing is open, click Switch To Source Drawing to make it the active drawing.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Open Source Drawing
OR
Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Open Source Drawing (Read-only)

OR

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Switch To Source Drawing

## Checking Out Project Object Source Drawings

You can check out a project object's source drawing at the project object level in the Prospector tree.

When you check out a drawing, a local copy of the drawing is placed in your working folder, and the drawing is opened.

**To check out a project object's source drawing**

1   In Toolspace, on the Prospector tab, expand the Projects collection.

2   Expand the project collection that contains the project object.

3   Expand the object collection that contains the project object.

4   Right-click the project object and click Check Out Source Drawing.

5   In the Check Out Drawing (page 2198) dialog box, specify whether file dependencies should be included.

6   Specify whether the latest version of the file is desired.

7   If Include File Dependencies is selected, specify which project files should be checked out with the drawing file by selecting the check boxes next to them.

8   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ <project> collection ➤ <object> collection ➤ right-click <object> ➤ Check Out Source Drawing

Dialog Box

Check Out Drawing Dialog Box (page 2198)

## Resetting a Checked Out Source Drawing

You can undo the checkout of a project object's source drawing at the project object level in the Prospector tree.

**To undo the checkout of a project object's source drawing**

1   In Toolspace, on the Prospector tab, expand the Projects collection.

2   Expand the project collection that contains the project object.

3   Expand the object collection that contains the project object.

4   Right-click the project object and click Undo Check Out Source Drawing.

5   In the Undo Check Out (page 2209) dialog box, select Include File Dependencies to display dependent files.

6   If dependent files are displayed, clear the check boxes next to files that should remain checked out.

**7** Click OK.

### Quick Reference

## Promoting Project Objects

You can replace a read-only object reference with an editable copy of the project object by promoting the object reference.

**WARNING** Promoting an object reference severs all connections to project object source data. The connection cannot be re-established.

You should promote an object reference only when you have no need to maintain the project object's link to its source drawing or data.

**To promote a project object**

**1** In Toolspace, on the Prospector tab, under the Open Drawings collection, expand the drawing that contains the project reference object you want to promote.

**2** Expand the drawing object collection that contains the object reference you want to promote.

**3** Right-click the object and click Promote.

### Quick Reference

# Working with Vault Project Points

Project management for AutoCAD Civil points differs from project management for other project objects.

Master copies of points are stored in a point database, not a source drawing, and points are checked out or copied directly to a drawing. The project points in a drawing are usually a subset of the project database. Individual points are not displayed in the project database, and they cannot be viewed using the Autodesk Vault tool.

Project points are accessed and managed through two files that are stored in the project database with other project files: *PointsGeometry.mdb*, which is the project point database, and *PointsStatus.mdb*, which contains information about the status (for example, checked-in or checked-out) of individual points. You do not explicitly check out or check in the project point database. Instead, you check out, modify, and check in individual project points and the project point database is updated automatically.

The status of your local project points in relation to the latest version of the project point database in the project's Vault database is shown by the icon displayed next to the project Points node under the project collection. The icons that can be displayed are shown in the following table:

| Icon | Meaning |
|---|---|
| | If no icon is displayed, the project point database exists in the project's Vault database, and you have not accessed it. |
| ○ | The local project point database matches the latest version of the project point database in the project's Vault database. |
| ● (green) | The local project point database is newer than the latest version of the project point database in the project's Vault database. |
| ● (red) | The local project point database is older than the latest version of the project point database in the project's Vault database. |

Many of the project management operations (such as check-in and check-out) that can be performed on points in a drawing can also be performed using point groups.

## Project Point State Icons

Because Vault is not used for managing individual points, the icons indicating the state of individual points (for example, checked-in or checked-out) are different from the icons used for other project objects.

The following table shows the project point state icons, which are displayed in the Points list view.:

| Icon | Meaning |
|---|---|
| 🔓 | The point is checked in. |
| ☑ | The point is checked out to your active drawing. |
| ☑ | The point is checked out to a drawing other than your active drawing. This could be a drawing that belongs to you or to someone else. |
| 🔒 | The point is protected and cannot be changed. |

## The Project Points List View

You can use the project points list view to obtain information about the state of project points.

**NOTE** To view project points in the Prospector tree, select Master View from the list at the top of the Prospector tab.

In Prospector ➤ Projects, click the project Points collection to display a list view that lists all of the project points. Project management information is listed, including:

■ Most recent version number

■ User who last modified the point and the date it was modified

■ Current state of the point (checked-in, checked-out, or protected) and the date the state was last changed

- User who set the current state of the point

- If checked-out, the drawing and host computer that the point is checked out to

Project point state icons are displayed next to each point. For more information, see Project Point State Icons (page 167).

For more information about changing the list view display, see Customizing a List View (page 84).

## Working with Point-related Project Management Commands

You can perform point access control operations, such as checking in and checking out points, in several locations in the Prospector tree.

When you select multiple points and perform a project management operation on them, the operation is performed only on those points to which it applies. For example, if you select a list of points in the Points list view and use the Check In command to check them in, only the points you checked out are checked in.

In addition, Survey points that are created using the Prospector Survey tab are not included in project management commands even if they are selected. For example, any Survey points included in the selection list during the Add to Project command are not added to the project.

You can access point-related project commands using list view shortcut menus or using point groups, as described below:

- If you already have a copy of a project point in a drawing, use the shortcut menu in the Prospector drawing Points list view to check in, check out, protect, or get a copy of the point.

- If you do not have a copy of the point in a drawing, use the shortcut menu in the Prospector project Points list view to check in, check out, protect, or get a copy of a point.

- If you have point groups defined in the drawing, right-click a point group in the Prospector tree and use the point group shortcut menu to check in, check out, protect, or get copies of project points.

In some cases, only valid project management commands are displayed on a shortcut menu. For example, if a set of points are not available for check out, the Check Out command might not be displayed, but the Get From Project command would be displayed so that you can get a read-only copy of the point. In some cases, a drawing must be saved before project management commands are available.

## Adding Points to a Project

You can use the Add To Project command to add points to a project so they can be accessed by others.

You can add some or all of the points in a drawing to a project. Before you can add drawing points to a project, you must save the drawing. If the drawing has not been saved, the Add to Project command is not available.

If the drawing contains one or more project objects, the points are added to the project that the existing project objects belong to. If the drawing does not contain project objects, you are prompted to select a project.

You can access the Add To Project command not only from the drawing's Points list view, but also from a drawing point group's shortcut menu.

**To add points to a project**

1  In Toolspace, on the Prospector tab, under the Open Drawings collection, expand the drawing that contains the points you want to add to the project.

**2** Click the Points collection to display the Points list view.

**3** In the list view, select the points, right-click, and click Add To Project.

Any Survey points included in the selection are not added to the project. Survey points are stored only in the project's Survey Point database.

**4** If you are prompted, select a project to add the points to.

**5** In the Add To Project (page 2195) dialog box, enter a comment.

**6** From the Check In Options list, select an option to specify the state of the points after you add them to the project.

**7** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Open Drawings collection ➤ click Points collection ➤ right-click selected points in list view ➤ Add To Project
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Add Points To Project

Dialog Box

Add Points To Project Dialog Box (page 2195)

## Checking Out Points

You can use the Check Out command to prevent others from making changes to project points while you are editing them.

To edit a project point, check it out to a drawing, edit it in the drawing, and then check it back into the project.

---

**NOTE** When working with project points, the term *local copy* refers to the copy of the point in your drawing.

---

Checking out a project point creates a local copy of the point in the drawing. The Check Out command gives you exclusive access to the project point; no one else can check it out while you are working on it.

If the Check Out command is not available, you can use the Get From Project command to a read-only local copy of the point. For more information, see Copying Project Points into a Drawing (page 171).

**To check out project points**

**1** Verify that the drawing you want to bring the points into is the active drawing. If it is not, right-click the drawing name in the Open Drawings collection in the Prospector tree and click Switch To.

**2** In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project that contains the points you want to check out.

**3** Click the project Points collection to display a list view.

**4** In the list view, select the project points you want to check out, right-click, and click Check Out.

**5** In the Check Out (page 2199) dialog box, click OK.

Local copies of the points are created in the active drawing.

### Quick Reference

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Check Out
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Check Out Points

Dialog Box

Check Out Points Dialog Box (page 2199)

## Checking In Points

You can use the Check In command to check in checked-out project points.

**NOTE** When working with project points, the term *local copy* refers to the copy of the point in your drawing.

After you edit one or more project points, use the Check In command to replace the existing project point with your local copy of the project point, and increase the project point's version number.

The Check In command always updates the project point from the drawing that the point was checked out to.

**To check in project points**

1   Verify that the drawing that you want to check the points in from is the active drawing. If it is not, right-click the drawing name in the Open Drawings collection in the Prospector tree and click Switch To.

2   Click the drawing's Points collection to display a list view.

3   In the list view, select the points to check in and right-click.

4   Click Check In.

5   In the Check In (page 2198) dialog box, enter a comment.

6   From the Check In Options list, select an option to specify the state of the points after you check them in.

7   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Check In
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Check In
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Check In Points

Dialog Box

Check In Points Dialog Box (page 2198)

## Copying Project Points into a Drawing

You can use the Get From Project command to make a read-only copy of project points in a drawing.

**To copy project points into a drawing**

1   Verify that the drawing to which you want to copy is the active drawing. If it is not, right-click the drawing name in the Prospector tree and click Switch To.

2   In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project collection that contains the points you want to copy.

3   Click the project Points collection to display the list view.

4   In the list view, select the points to copy, right-click, and click Get From Project.

5   In the Get From Project (page 2204) dialog box, click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Get From Project
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Get From Project
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Get Points From Project

Dialog Box

Get Points From Project Dialog Box (page 2204)

## Protecting Project Points

You can use the Protect command to prevent project points from being checked out and edited.

You cannot protect a project point that is checked-out.

**To protect project points**

1   In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project collection that contains the points you want to protect.

2   Click the project Points collection to display the list view.

3   In the list view, select the points to protect, right-click, and click Protect.

4   In the Protect (page 2207) dialog box, click OK.

A lock icon 🔒 is displayed next to a protected project point in the Points list view.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Protect
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Protect
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Protect Points

Dialog Box

Protect Points Dialog Box (page 2207)

## Unprotecting Project Points

You can use the Unprotect command to allow protected project points to be checked out and edited.

Only the person who protected a project point can unprotect it.

A lock icon 🔒 is displayed next to a protected point in the Points list view.

**To unprotect project points**

1   In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project collection that contains the points you want to unprotect.

2   Click the Points collection to display the list view.

3   In the list view, select the protected points, right-click, and click Unprotect.

4   In the Unprotect (page 2209) dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Unprotect
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Unprotect
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Unprotect Points

Dialog Box

Unprotect Points Dialog Box (page 2209)

## Deleting Project Points

You can use the Delete From Project command to delete project points.

When you delete a project point, it is permanently removed from the project, and is no longer listed in the Prospector tree.

Before you can delete a project point, you must check it out. For more information, see Checking Out Points (page 169).

**To delete project points**

1   In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project collection that contains the points you want to delete.

2   Click the Points collection to display the list view.

3   In the list view, select the points to delete, right-click, and click Delete From Project.

4   In the Delete From Project  (page 2204)dialog box, click OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Delete From Project
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Delete From Project
OR
Prospector tab: Open Drawings collection ➤ Point Groups collection ➤ right-click <point group> ➤ Delete Points From Project

Dialog Box

Delete Points From Project Dialog Box (page 2204)

## Resetting Points You Checked Out

You can use the Undo Check Out command to reset checked-out project points back to a checked-in state.

Use this command if you checked out one or more points but do not want to update the project with any changes you have made.

You can use this command only on project points that are checked out to you.

**To reset your checked out project points**

1   In Toolspace, on the Prospector tab, expand the Projects collection and then expand the project collection that contains the points you want to reset.

2   Click the Points collection to display the list view.

3   In the list view, select the points to reset, right-click, and click Undo Check Out.

4   In the Undo Check Out (page 2209) dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Undo Check Out
OR
Prospector tab: Open Drawings collection ➤ drawing Points collection ➤ right-click selected points in list view ➤ Undo Check Out
OR
Prospector tab: Open Drawings collection ➤ right-click <point group> ➤ Undo Check Out

Dialog Box

Undo Check Out Points Dialog Box (page 2209)

## Resetting Project Points Set by Others

If you are a system administrator, you can use the Reset To Checked In command to reset to a checked-in state a project point checked-out or protected by someone else.

Use this command if a point is checked out or protected and for some reason the person who originally checked it out or protected it cannot undo the operation or check in the point.

**NOTE** The Reset to Checked In command is available only to a system administrator.

**To reset points checked out to someone else**

1 In Toolspace, on the Prospector tab, expand the Projects collection, and then expand the project collection that contains the points you want to reset.

2 Click the Points collection to display the list view.

3 Select the checked-out or protected points, right-click, and click Reset To Checked In.

4 In the Reset To Checked In (page 2207) dialog box, click OK.

**NOTE** The Project Points item list can be sorted to assist selecting the project points to be reset to a checked in state.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab (Master View): Projects collection ➤ project Points collection ➤ right-click selected points in list view ➤ Reset to Checked In

Dialog Box

Reset Points To Checked In Dialog Box (page 2207)

# Error Codes

Errors encountered during Vault project management commands are reported in two ways.

If a single error is encountered during a project management operation, it is displayed in a message box. If multiple errors occur during a single operation, the resulting Vault error codes are displayed in the Event

Viewer. For example, if a drawing check-in operation fails, and then subsequent check-ins of dependent files also fail, the error codes for the related multiple failures are displayed in the Event Viewer.

The following table lists the Vault error codes that can be displayed if a project management command fails:

| Group Code/Error Code | Error Description |
| --- | --- |
| SysErrCodes | |
| 0 | Error code is not valid or is unspecified. |
| CoreErrCodes | |
| 100 | Error creating the knowledge vault in database. |
| 101 | Script to prime the database with its schema failed. |
| 102 | Knowledge vault already exists in database. |
| 103 | Attempt to create user in database failed. |
| 104 | Error adding user role in database. |
| 105 | Error removing user role in database. |
| 106 | Error validating the principal as being a secure principal (i.e. authenticated). |
| 107 | Principle requirement does not match. Should be anonymous. |
| 108 | Requires new transaction, but transaction was not created. |
| 109 | Error initializing knowledge vault in database. |
| 110 | Stored procedure cannot be found. |
| 111 | Error getting relative resource path. |
| 112 | File stream is null, so the crc code can't be calculated |
| 113 | File is not readable, so the crc code can't be calculated |
| 114 | Could not create (Knowledge)VaultMaster Database |
| 115 | Could not create (Knowledge)VaultMaster Database schema |
| 116 | Could not initialize (Knowledge)VaultMaster Database tables |

| Group Code/Error Code | Error Description |
| --- | --- |
| 117 | Failure during migrate of a KnowledgeVault |
| 118 | Failure creating VultSys user for the DB |
| 119 | Failure during migrate of KnowledgeVault-Master |
| 120 | Failure updating the KnowledgeVaultMaster stored procedures |
| 121 | Failure deleting the KnowledgeVaultMaster |
| 122 | Failure updating user information |
| 123 | Could not update ElDatabase table |
| 124 | Failure moving the file store |
| 125 | Failure attaching a database |
| 126 | Failure detaching a database |
| 127 | Failure deleting a Vault |
| 128 | Failure backing up |
| 129 | Failure restoring |
| 130 | Unable to determine the version of a KnowledgeVault or Master |
| 131 | The database master login is invalid |
| 132 | The directory is not empty |
| 133 | The Knowledge Vault referenced doesn't exist |
| 134 | There are Knowledge Vaults still attached. |
| 135 | Failure activating a vault |
| 136 | Failure deactivating vault. DB is probably in use |
| 137 | One of the inputs to the service call is incorrect. |
| 138 | Vault name is not allowed, most likely due to illegal characters. |
| 139 | Specified folder is illegal. |
| 140 | Specified folder is already in use. |

| Group Code/Error Code | Error Description |
| --- | --- |
| 141 | Could not find global option. |
| 142 | Could not find local option. |
| 143 | Duplicate User Name |
| 144 | Database error because an MDF or LDF file with that name already exists. |
| 145 | Database is in use |
| 146 | Cannot determine the migration steps. |
| 147 | The specified path is too long. |
| 148 | Incorrect vault category Id entered |
| 149 | The vault's category does not support the operation. |
| 150 | The KnowledgeVaultMaster referenced doesn't exist |
| 151 | Not enough disk space to perform Backup/Restore operation. |
| 152 | Cannot restore db files to a remote location |
| 153 | Selected directory does not contain a valid backup structure. |
| SecurityErrCodes | |
| 300 | Authentication Token is invalid. |
| 301 | User name and/or Password is invalid, so user cannot be authenticated. |
| 302 | User is not a member of the vault |
| 303 | Invalid permissions for transaction |
| 304 | User is disabled |
| 305 | Vault database does not exist |
| DocServiceErrCodes | |
| 1000 | Folder id is invalid. |
| 1001 | Cannot get the latest version of file. |

| Group Code/Error Code | Error Description |
| --- | --- |
| 1002 | Cannot get file version. |
| 1003 | File id is invalid. |
| 1004 | Checkout latest file version failed. |
| 1005 | Error checking in file version into database. |
| 1006 | Error undoing check out of file version. |
| 1007 | Bad version id when getting file version dependents or dependencies by version id. |
| 1008 | Cannot add file because file exists. |
| 1009 | Cannot add file (unspecified failure) |
| 1011 | Cannot add folder because folder exists. |
| 1012 | Cannot add folder (unable to create/make new folder). |
| 1013 | Cannot get file (file id is invalid). |
| 1014 | Cannot create/make version in database. |
| 1015 | Cannot delete file version because of dependencies. |
| 1016 | Cannot undo checkout because user is not the same as user who checked out file. |
| 1017 | Cannot undo checkout because passed in folder id is not the same folder that the file was checked out from. |
| 1018 | Cannot check in file because the file is not currently checked out |
| 1019 | Cannot check in file because the file is not currently checked out by the same user. |
| 1020 | Cannot check in file because passed in folder id is not the same folder that the file was checked out from. |
| 1021 | Cannot check out the file because it is already checked out. |
| 1022 | Circular dependency check failed. |
| 1023 | Cannot create folder in database. |

| Group Code/Error Code | Error Description |
| --- | --- |
| 1024 | Cannot get folder. |
| 1025 | Cannot get root folder from the database. |
| 1026 | File belongs to a library folder. |
| 1027 | Folder is a library folder. |
| 1028 | Cannot move file. |
| 1029 | A file exists in the destination folder that has the same name as the file that is being moved from its source folder. |
| 1030 | Cannot share because a file already exists in the destination folder with the same name. |
| 1034 | Cannot rename the file because there was some other unexpected error. |
| 1035 | Could not create property definition in database. |
| 1036 | Cannot get all property definitions. |
| 1037 | Cannot get all extended property definitions. |
| 1039 | Property could not be added in database. |
| 1040 | Big property could not be added in database |
| 1041 | Cannot get properties |
| 1042 | Create folder rule-check failed: parent must exist, for all but root |
| 1043 | Create folder rule-check failed: libs can only have non lib parent if that parent is root. Libs cannot have non-lib children. |
| 1044 | Request to share a file to a folder fails because of a concurrent request to share the file to the same folder. |
| 1045 | Request to move a file to a folder fails because of a concurrent request to move the file to the same folder or because of a concurrent request to move the file to another folder |

| Group Code/Error Code | Error Description |
| --- | --- |
| 1046 | Request to create a folder fails because the folder name is longer that 80 characters. |
| 1050 | Request to conditionally delete a file fails because there are delete restrictions (file has dependent parent files, file is checked out, or file is linked or attached to an item) |
| 1051 | Request to unconditionally delete a file fails because the are delete restrictions that cannot be overridden (file is linked or attached to an item) |
| 1052 | Request to delete a file failed for an unspecified reason. |
| 1053 | Request to conditionally delete a folder fails because there are delete restrictions on one or more child files (file has dependent parent files, file is checked out, or file is linked or attached to an item) |
| 1054 | Request to unconditionally delete a folder fails because the are delete restrictions that cannot be overridden on one or more child files (file is linked or attached to an item) |
| 1055 | Request to delete a folder fails due to an unspecified reason |
| 1056 | Occurs when either no parameters have been set and a purge is attempted or when an attempt is made set invalid parameter values. |
| 1057 | Occurs when something goes wrong while purging file iterations from the database, or while deleting files from the file store |
| 1058 | If the Unique File Name Required Vault option is ON, a request to Add or Check in a file with the same name as a file already existing in the Vault will fail with this error. |
| 1059 | Occurs when an attempt to update a Folder fails for an unspecified reason. |
| 1060 | Occurs when an attempt to update a Folder Name fails because another Folder with that name exists in the parent. |
| 1061 | Label ID is invalid |

| Group Code/Error Code | Error Description |
| --- | --- |
| 1062 | Label Name contains invalid characters |
| 1063 | Label Name already exists in vault |
| 1064 | Cannot create label in database. |
| 1065 | |
| 1066 | No PropertyGroup with the given Id exists |
| 1067 | A PropertyGroup with the specified name already exists. |
| 1068 | The PropertyDefinitions to be assigned to the PropertyGroup do not share the same DataType. |
| 1069 | The PropertyGroup must be assigned PropertyDefinitions |
| 1070 | Request to add a PropertyGroup failed for an unspecified reason |
| 1071 | Request to update a PropertyGroup failed for an unspecified reason. |
| 1072 | Request to delete a PropertyGroup failed for an unspecified reason. |
| 1073 | An unknown error occurred. |
| 1074 | Folder with the same name already exists in the destination folder |
| 1075 | Folder being moved has descendent files that are checked out. |
| 1076 | Move folder rule-check failed: parent must exist, for all but root |
| 1077 | Move folder rule-check failed: libs can only have non lib parent if that parent is root. Libs cannot have non-lib children. |
| 1078 | A null path or path with illegal characters has been passed in. |
| 1079 | The full path of the folder is greater than 256 characters |
| 1080 | A null value has been passed in where null values are not allowed. |

| Group Code/Error Code | Error Description |
|---|---|
| 1081 | The date is out of range for the DB. The date should be between 1-1-1753 and 12-31-9999 |
| SerializeErrCodes | |
| 1200 | Exception passed to Encodable Exception saying the object is null |

# Project Management Command Reference

Use these commands to access reference functionality for data shortcuts or Vault.

The following table lists AutoCAD Civil commands related to project management and briefly describes their functionality.

| Command | Description |
|---|---|
| CreateAlignmentReference | Creates a Vault alignment reference (page 893) |
| CreateNetworkReference | Creates a Vault pipe network reference (page 1199) |
| CreateProfileReference | Creates a Vault profile reference (page 1055) |
| CreateSurfaceReference | Creates a Vault surface reference (page 609) |
| CreateViewFrameGroupReference | Creates a Vault view frame group reference (page 1709) |
| CreateDataShortcuts | Creates a data shortcut for each eligible object in the active drawing (page 131) |
| NewShortcutsFolder | Creates a new project folder for use with data shortcuts (page 129) |
| PromoteAllReferences | Promotes all reference objects in the drawing, whether they are Vault or data shortcut references (page 134) |
| PromoteNetwork | Promotes a data shortcut reference pipe network (page 134) |
| PromoteReference | Promotes a data shortcut reference surface, alignment, or profile (page 134) |
| PromoteViewFrameGroup | Promotes a data shortcut reference view frame group (page 134) |
| RepairBrokenReferences | Repairs one or more data shortcut references in a consumer drawing (page 135) |
| SetShortcutsFolder | Changes the active project folder for use of data shortcuts (page 130) |

| Command | Description |
| --- | --- |
| SetWorkingFolder | Points to a different working folder for data short-cuts projects (page 129) |
| ShortcutNode | Hides or shows the Data Shortcuts node on the Prospector tab (page 133) |
| SynchronizeReferences | In data shortcuts projects, updates data references with current versions of their source objects (page 133) |
| ValidateShortcuts | Ensures that all data references are up to date and reloads any outdated ones from their source objects (page 135) |

# Survey

# Understanding Survey

<div style="text-align: right; font-size: large;">**8**</div>

You can use the AutoCAD Civil survey features to download, create, analyze, and adjust survey data. The survey features extend the functionality of AutoCAD Civil by streamlining the process of transferring field-captured survey data to and from the office.

Survey Best Practices

## Survey Overview

You can use the complete set of tools in AutoCAD Civil to import survey data, perform surveying calculations, and automate symbol placement and line work.

AutoCAD Civil survey data and figures persist in a survey database, and can be accessed using multiple drawings attached to the same AutoCAD Civil survey database.

The survey database stores angle values in Radians and distance values in Meters. Survey data is transformed according to the database units for the purposes of display and input. AutoCAD Civil drawing units can be independent of the survey database units. If drawing objects are imported from the survey database and the AutoCAD Civil drawing units and coordinate zone differ, the objects are transformed.

Survey objects and their components are displayed based on their styles. Before creating survey components, you should be familiar with creating and managing styles (page 53).

### Establish Survey Settings

Define and manage the following survey settings:

| | |
|---|---|
| Figure Prefix Database | Determine the display and properties of figures. |
| Survey Equipment Database | Establish an error model for a specific survey instrument when analyzing survey data using least squares. |
| Survey Database Settings | Control the default settings for all database components. |
| User Settings | Control the default settings for survey specific components. |

| Linework Code Set | Define the codes used to determine the connectivity of linework from point descriptions. |
| --- | --- |
| Survey Feature Settings | Control the default styles for figures and networks. |

For more information about settings, see Survey Settings (page 205).

**Import and Edit Survey Data**

You can import and modify survey data in the AutoCAD Civil environment and use the AutoCAD Civil drawing session to graphically display the survey data. Use the AutoCAD Civil survey data to affect other AutoCAD Civil data, such as points and surfaces.

| Survey Data Collection Link | Convert raw survey observation data to a field book file (*.fbk*) that can be imported into an AutoCAD Civil survey database. |
| --- | --- |
| Import Survey Data Wizard | Use the Import Survey Data wizard to import field book files, point files, and LandXML files or to select points in the drawing. |
| Import Commands | Use the commands available from a network to import a field book, point file, or points in a drawing. |
| Import Survey Land XML | Import Survey LandXML data into the survey database. |
| Import Events | An import event is created each time a data source is imported into the survey database. Use import events to process linework and re-import data with existing settings, also to delete only the data created from the import event. |
| Add survey data to a network | Use the commands from within a network collection to create or edit control points, non-control points, directions and observations from a network. |
| Survey Command Window | Perform coordinate geometry operations using specific survey commands, along with the options to record input and output, and to re-run the input from a batch file. |

For more information about input and figures, see Adding and Editing Survey Data (page 253) and Survey Figures (page 321).

**Perform Analysis Adjustments**

Using traverse and least squares adjustments, you can reduce field data for analysis.

| | |
|---|---|
| Define a traverse | Perform Compass, Crandall, Transit, and Least Squares analysis. |
| Least Squares analysis | Perform 2D or 3D network least squares analysis. Update points in the survey database and the linework input. |
| Mapcheck Analysis | Check the closure of values from label objects based on the precision of the annotation of the label object, or enter mapcheck data manually. |

For more information, see Survey Analysis and Output (page 361).

**Export Survey Data**

You can output and export survey data:

| | |
|---|---|
| Export survey data | Export data as a field book or Survey LandXML file. |
| Define surface breaklines | Use survey figures as breaklines when you create a surface model. |
| Update the survey objects in the drawing | Select a survey object in the drawing and use short-cut menus to update the drawing object on the Prospector tab. |

# Survey Project Phases

Organize survey data by creating networks within the survey database for the various stages of a project.

Throughout a survey project it may be necessary to perform several types of survey tasks such as a site analysis, a boundary analysis, and a topographic survey.

In AutoCAD Civil you can create multiple networks within a survey database and append survey data to the survey database to maintain the integrity of existing survey data. The benefits of multiple networks are:

■ Each network can represent a specific purpose.

■ Each network can be analyzed and adjusted separately, but can reference data from another network.

■ When you create figures by importing data into a network, that network contains information as to which figures will need to be updated if the network is modified.

# Site Analysis Phase Workflow

In the first phase of a project, a preliminary analysis of a site is necessary to determine if the project is feasible.

As an example of this phase, a survey crew would go to the site with a mapping grade GPS data collector to map wetlands that have been delineated on the site. This information is imported into a survey network of the project's survey database and overlaid onto a map of the area along with other relevant information, such as a steep slope analysis of the site's topography.

To accomplish this phase in AutoCAD Civil:

| | |
|---|---|
| **Create a new project** (page 128) | In Toolspace on the Prospector tab choose Master View and create a new project or create a local survey database. |
| **Check out the Survey Database** (page 125) | If you are working with the project management tools, check out the Survey database and import the wetland mapping points and line features. |
| **Create a new survey network** (page 201) | Create a network with a relevant name to the site (<name>_Site Analysis) and import the points and line work into the survey database. |

## Boundary Survey Phase Workflow

In the second phase, a boundary survey is performed to locate physical evidence and establish horizontal and vertical control on the site.

After determining that a site development is feasible and deed research is completed, a survey crew performs a boundary survey. Survey grade GPS control is established on the site to provide a tie onto the State Plane coordinate system. When the boundary survey is completed, the data (survey observations and linework) is imported into another survey network in the same project survey database. The data is analyzed to determine positional error and the network is adjusted.

To accomplish this phase in AutoCAD Civil:

| | |
|---|---|
| **Create a new survey network in the project database** (page 201) | Create a network with a relevant name (<name>_Boundary Survey). |
| **Import the field book file** (page 241) | Import the survey data into the survey database. |
| **Perform traverse and least squares analysis** (page 361) | Determine error of closure and positional accuracy and adjust the survey data. |
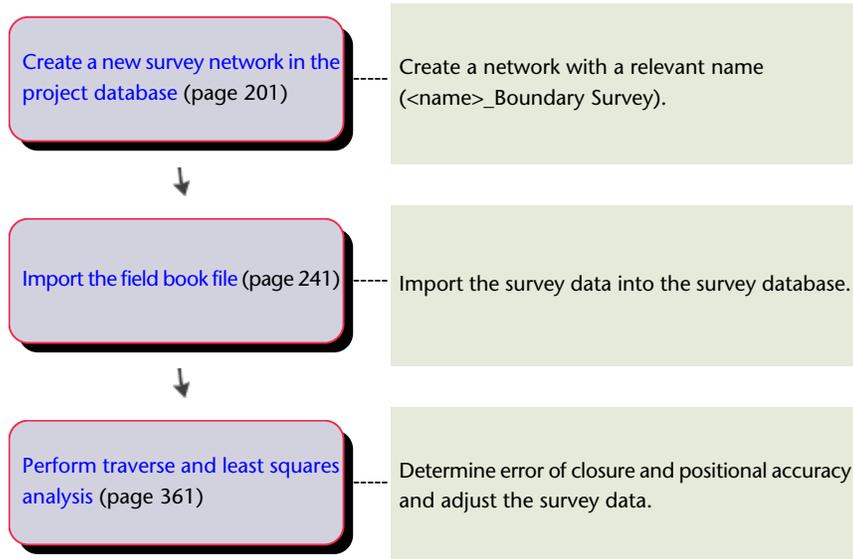
# Boundary Analysis Phase Workflow

In the third phase, the boundary is closed and a survey figure is created.

To maintain the integrity of the boundary survey network, you can create a new network and perform iterative coordinate geometry calculations. The parcel can be closed by creating a survey figure from the coordinate geometry calculations in conjunction with the points located in the field. You can query the survey figure to report the figure inverse and area. The figure can then be inserted into a drawing as the basis of a new subdivision.

To accomplish this phase in AutoCAD Civil:

| | |
|---|---|
| Create a new survey network in the project survey database (page 201) | Create a new network with a relevant name (<name>_Boundary Analysis). |
| ↓ | |
| Perform coordinate geometry calculations on the network (page 2419) | In the Survey Command Window, use the Survey Command Language, commands to close the parcel and create a survey figure by performing coordinate geometry calculations. |
| ↓ | |
| Mapcheck (page 356) and Figure Inverse (page 357) | Query the figure to report mapcheck and figure inverse. |
| ↓ | |
| Insert the figure into a drawing (page 324) | The new figure can be used a basis in creating a subdivision layout. |

# Topographic Survey Phase Workflow

In the last phase, a topographic survey of the proposed road locations is performed.

The topographic survey supports the engineering of the street grades, upgrade of the access from existing streets, site drainage, and driveway access to the proposed building sites. The topographic survey is based on the horizontal and vertical control established from the boundary survey network. After completing the topographic survey, the data is imported into a new survey network. This survey network references control from the boundary survey, and creates additional survey points and linework that can be used to create a terrain model for the site engineering.

To accomplish this phase in AutoCAD Civil:

| | |
|---|---|
| Create a new survey network (page 201) | In the survey database, create a new network with a relevant name (<name>_Topo). |
| ↓ | |

| | |
|---|---|
| **Import the topographic survey data** (page 241) | Import topographic survey data into the survey network using the Import Field Book command. |
| ↓ | |
| **Create a new drawing** (page 114) | Create a new drawing that contains the surface for the site engineering. |
| ↓ | |
| **Create a point group and add the survey points** (page 529) | Create a new point group and add the survey points. |
| ↓ | |
| **Create a new surface** (page 601) | Create a new surface and add the point group to the surface. |
| ↓ | |
| **Add the survey figures as breaklines** (page 324) | Add the survey figures (located in the topographic survey) to the surface as breaklines. |
| ↓ | |
| **Edit the surface** (page 611) | Perform additional edits on the surface, such as editing the surface boundary. |
| ↓ | |
| **Working with Vault Project Drawings** (page 156)**Adding a Drawing to a Project** (page 156) | Add the drawing with the new surface to the project if you are using project management. |

## Example: Using Networks with Import Events and Survey Data

The following descriptions of survey phases show an example of using networks and import events with survey data.

**Phase One**

■ A survey is being done on a parcel of land by multiple survey crews. The survey will be analyzed within a single survey network, for example a least squares network analysis.

- The data from each survey crew is downloaded into separate *.fbk* files and each *.fbk* file is imported into each network, creating a seperate import event. During each import, potential point identifier (point name or point number)conflicts can be resolved by applying a point identifier offset, and linework can be processed using the specified linework code set.

- The network is analyzed creating adjusted control points for each setup and updated coordinates for each sideshot within each setup. The coordinates for each figure are updated in each import event.

**Phase Two**

- In a later stage of the civil engineering process, an additional survey has to be performed to establish control for construction stakeout of proposed roads and parcel corners.

- This additional survey ties into the network that was adjusted in phase one and also requires a least squares analysis.

- A new network is created, and for each survey data file that is imported into this network an import event is created.

# Survey User Interface

Manage, analyze, and edit survey data using the Toolspace Survey tab, the Survey collection in the Prospector tab, the Survey collection in the Settings tab, the Survey Command Window, and various panorama vistas.

## Survey Tab

Use the Survey tab in Toolspace to access and manage survey settings, survey-related databases, and survey project data.

For more information about the Survey tab, see The Toolspace Survey Tab (page 97).

## Survey Collection (Prospector Tab)

Use the Survey collection on the Prospector tab in Toolspace to access survey data contained in the drawing. As you insert survey network and figure objects into the drawing, they are displayed in the Prospector Survey collection.

Expand the Survey collection to view the names of the survey components and display a tabular list of the components in the Prospector list view. For more information, see The Toolspace Item View (page 82).

Since the Prospector tab displays only the drawing representations of survey data, you manipulate and edit the data in the Survey tab.

**To browse to survey data from the Prospector tab**

- In Toolspace, on the Prospector tab, right-click a network object or a figure object ➤ Browse To Survey Data.
  On the Survey tab, the survey tree expanded, displaying the selected network or figure.

**To update a network or figure drawing object from survey data**

- In Toolspace, on the Prospector tab, right-click a network object or a figure object ➤ Update From Survey Data.
  The drawing object is updated with the data in the survey database.

**To update survey data from the drawing**

■ In Toolspace, on the Prospector tab, right-click a figure object ➤ Update Survey Data From Drawing. The survey database is updated with changes made to the selected figure object.

## Survey Collection (Settings Tab)

Use the Survey collection in the Settings tree to manage survey drawing settings and survey styles.

Right-click the Survey collection to edit survey feature settings. For information about survey feature settings, see Edit Feature Settings - Survey Dialog Box (page 2373).

Expand the Survey collection to display and edit the styles that are available for survey objects. For information, see Survey Styles and Display (page 195).

## Survey Command Input

Use the Survey Command Window to access survey input and analysis functionality with menu choices, or you can enter survey commands directly.

For more information about the Survey Command Window layout and options, see Survey Command Window (page 2419).

### Survey Command Language

The survey command language (which is also used in field book files) is one way to enter survey data into AutoCAD Civil.

Commands entered at the survey command line use a specific syntax. For information about the syntax and available commands, see Survey Command Reference (page 391).

**To display the Survey Command Window**

■ In Toolspace, on the Survey tab, expand the Network collection. Right-click a network item. Click Survey Command Window.

### Quick Reference

Toolspace

  Survey tab: ➤ right-click <named network> ➤ Survey Command Window

Dialog Box

  Survey Command Window (page 2419)

## Survey Objects

Survey networks and figures persist in AutoCAD Civil drawings as objects with the names AeccSvNetwork, AeccSvFigure, AeccDbSvNetwork, and AeccDbSvFigure.

■ Network object. Represents a survey network or traverse in the drawing. The network object is a read-only object that cannot be edited in the drawing. If the network is modified in the database, the survey network displayed on the Survey tab in Toolspace indicates that it needs to be updated.

■ Figure object. Represents a survey figure in a drawing.

The figure object can be edited in the drawing (depending on how the figure is defined), but it displays as out-of-sync with the survey database. The survey database can be updated from the figure drawing object. If other drawings have the same figure in a drawing and the survey database has been updated, their drawing figures are shown as out-of-date and can be updated from the survey database.

When a figure with a Lot Line property is inserted into the drawing, an AutoCAD Civil site is created in the drawing (if it does not exist) and Aecc_Parcel_Segment objects are created in the site that is specified for the figure. If the figure is closed, a parcel object of the same geometry is added. If not, then parcel segments are added to the site for the figure geometry.

■ **Survey Point**. A special type of COGO Point that is read-only in a drawing session. However, the display properties of a Survey Point can be modified in the Survey points collection on the Survey tab in Toolspace, or using the Edit Survey Point Properties command. A survey point is read-only in the drawing, because a survey point has dependent data, such as an Angle, Distance, Vertical Angle, Prism Height, from a specific instrument setup.

Survey drawing objects are displayed in the Toolspace Prospector tab under the Survey collection.

For general information about AutoCAD Civil objects, see Understanding Objects and Styles (page 51).

# Survey Styles and Display

Use the survey styles to control the way that survey networks and figures are displayed in a drawing.

The survey styles are accessed and managed on the Settings tab in Toolspace, where you can create, edit, copy, and delete them. For more information, see The Object Style Collection (Settings Tree) (page 94).

There are two survey-related styles:

■ Network styles. Include display control for survey network components, including known and unknown control points, non-control points, sideshot lines and sideshot points, direction lines, network lines, error ellipses, and tolerance error points. You can also control the network marker styles and 3D geometry display. For more information, see Survey Network Style Dialog Box (page 2388).

■ Figure styles. Include display control for survey figure components, including plan and model markers, and 3D display. For more information, see Survey Figure Style Dialog Box (page 2390).

For more general information about styles, see Object Styles (page 53).

**To create a network style**

1  In Toolspace, on the Settings tab, expand the Survey collection. Right-click the Network Styles collection. Click New.

2  In the Network Style dialog box, click the Information tab and enter a name and description for the style.

3  To specify the styles for markers and the error ellipse scale factor in the network, click the Components tab. For information on the components about this tab, see Components Tab (Survey Network Style Dialog Box) (page 2389).

4  To specify the 3D geometry display of the network, click the 3D Geometry tab and specify the settings.

5  To define the display of the various network components, including network lines, points, and error ellipses, click the Display tab.

6  To view summary information about the style, click the Summary tab.

7  Click OK.

**To create a figure style**

1 In Toolspace, on the Settings tab, expand the Survey collection, right-click the Figure Styles collection, and click New.

2 In the Figure Style dialog box, click the Information tab and enter a name and description for the style.

3 To define figure marker placement and styles, click the Markers tab and specify the options. For information about the components on this tab, see Plan and Model Tab (Survey Figure Style Dialog Box) (page 2391).

4 To specify the 3D geometry display of the figure, click the 3D Geometry tab and specify the settings.

5 To define the display of the various figure components, including figure lines, and markers, click the Display tab.

6 To view summary information about the style, click the Summary tab.

7 Click Apply.

**To edit a survey style**

1 In Toolspace, on the Settings tab, right-click the name of the network or figure style that you want to edit and click Edit.

2 In the Style dialog box, change the properties of the style as required.

# Survey Databases

In AutoCAD Civil, survey data is not drawing dependent and is stored in an external database. For display and visualization, survey data can be manually and automatically inserted into a drawing when the survey database is updated or when data is imported into the survey database.

The Survey *.sdb* file is the main survey database and it contains all the data in the survey database collections except for the Extended Properties definitions and values. The Survey *.sdx* file contains the Extended Properties definitions and values.

# Survey Database

Use the Toolspace Survey tab to create local survey databases. You can subsequently create a new AutoCAD Civil Project from the existing local survey database.

A survey database contains all the control points, known directions, observation measurements, traverse definitions, figures, and standard deviations based on equipment data for the survey database. This includes observations imported from data collector files, entered from the Toolspace Survey tab, the survey editors (for example, Traverse Editor and Observation Editor), and Survey Command Window.

A survey database is displayed on the Toolspace Survey tab under the Survey Databases collection.

By default, survey databases are local and do not use the AutoCAD Civil project management functionality. You can subsequently create a new AutoCAD Civil Project from the existing local database. When a new AutoCAD Civil project is created from Prospector, a new Survey database is automatically created and must be checked out to add or modify data.

**NOTE** AutoCAD Civil project management determines the working folder, therefore the databases that are displayed in the working folder may vary depending on whether you are logged in or logged out.

For more information about AutoCAD Civil project management, see Project Management (page 125).

## Using a Local Survey Database

Use the Toolspace Survey tab to create local survey databases.

Each local survey database is stored in the project working folder, for example: *Civil Projects\<project name>*. You can set the current working folder from either the Survey Databases collection on the Survey tab or in the Projects folder on the Prospector tab if you have Vault Explorer installed.

A survey database is displayed on the Toolspace Survey tab as a named survey database item under the Survey Databases collection.

**To create a local survey database**

1   In Toolspace, on the Survey tab, right-click the Survey Databases collection.

2   Click New Local Survey Database.

3   In the New Local Survey Database dialog box, enter the name for the database.

   The database is created and displayed in bold text under the Survey Databases collection.

**To change the location for local survey databases**

1   In Toolspace, on the Survey tab, right-click the Survey Databases collection.

2   Click Set Working Folder.

   **NOTE** If you have Vault Explorer installed you can also set the working folder on the Prospector tab in the Projects collection.

3   In the Browse For Folder dialog box, browse to the working folder location.

   Survey databases in the working folder are displayed in the Survey Databases collection and all local survey databases that you subsequently create are stored in this location.

**NOTE** By default, to save on resource usage, when you start AutoCAD Civil, all survey databases are displayed in a closed state.

**To open or close a survey database**

1   In Toolspace, on the Survey tab, right-click the named survey database. Click Open Survey Database or double click the database in the collection.

   The named survey database is displayed in bold text and it is expanded to display the Networks, Network Groups, Figures, Figure Groups, Survey Points, and Survey Point Groups collections.

   **NOTE** Only one survey database can be open at a time.

2   To close an open database, in Toolspace, on the Survey tab, right-click the named database. Click Close Survey Database.

## Quick Reference

Ribbon

Click Home tab ➤ Palettes panel ➤ Open Survey Toolspace 🛰

Toolspace

Survey tab: expand Survey Databases ➤ right-click <database name> ➤ Open Survey Database

or

Survey tab: expand Survey Databases ➤ right-click <database name> ➤ Close Survey Database

## Adding a Survey Database to a New Project

Create a new AutoCAD Civil project from an existing survey database to take advantage of the standard project management functionality when accessing survey databases.

After you add a survey database to a project, you must check it out to make changes to the survey data.

For more information about AutoCAD Civil project management, see Project Management (page 125).

**To create a new AutoCAD Civil project from an existing survey database**

1   In Toolspace, on the Survey tab, right-click the survey database. Click Add To New Project.

   **NOTE** The Add To New Project menu item is displayed if you have logged into Vault. For information, see Logging in to a Vault Server (page 149).

2   In the Add To New Project dialog box, the database name is displayed. The project icon is displayed next to the survey database name to indicate that the survey database is contained in an existing AutoCAD Civil project.

3   In Toolspace, click the Prospector tab, expand the Projects collection and right-click the newly created project to access the standard project management functions. See Working with Vault Projects (page 148).

   **NOTE**   If the Projects collection is not visible, click Master View from the list.

### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ right-click <database name> ➤ Add To New Project

## Translating a Survey Database

Use this command to perform a simple translation of a selected survey database.

This command is useful when you need to move all the data in the survey database from an assumed location to a known location, subsequent to importing the survey data into AutoCAD Civil.

Translate the selected database by specifying a base point, rotation angle, destination point, and optional elevation.

**To translate a survey database**

1   Click Survey tab ➤ Modify panel ➤ 🔻 ➤ Translate Database.

2   On the Translate Survey Database Base Point page, enter a point number from the open database or click Pick In Drawing and select a point in the drawing. Click Next.

   **NOTE** If the base point exists as a survey point within the survey database, the Name, Easting, Northing, Elevation, Description, Latitude, and Longitude property values are displayed.

**3** On the Translate Survey Database Rotation Angle page, click the value cell and enter a rotation angle. Click Next.

**4** On the Translate Survey Database Destination page, click the value cell and enter Easting, Northing, and optional Elevation Change values. Click Next.

**5** On the Translate Survey Database Summary page, review the values for Base Point, Destination Point, and Translation. Click Finish.

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Translate Database

Menu

Survey menu ➤ Translate Survey Database

Toolspace

Right-click <database name> ➤ Translate Survey Database

Command Line

TranslateSvDatabase

## Survey Equipment Database

The survey equipment database contains one or more equipment definitions.

Equipment definitions specify the values associated with a specific surveying instrument, such as the standard deviations associated with the measuring capabilities.

For information on equipment definition settings, see New Equipment/Equipment Properties Dialog Box (page 2384).

**To create an equipment database and definition**

**1** In Toolspace, on the Survey tab, right-click the Equipment Databases collection. Click New.

**2** In the New Equipment Database dialog box, enter a name for the new equipment database. The database is created with the default equipment definition named 'Sample,' which is displayed in the Equipment Databases collection.

**NOTE** A best practice is to enter the model number or the name of the equipment manufacturer in the Name field.

**3** To create a new equipment definition, right-click the equipment database and click New.

**4** In the Equipment Properties dialog box, enter a name for the equipment definition and modify other properties as required. For more information about the equipment properties, see New Equipment/Equipment Properties Dialog Box (page 2384).

**To make an equipment definition current**

■ In Toolspace, on the Survey tab, expand the Equipment Databases collection. Expand the named equipment database. Right-click the equipment definition. Click Make Current.

**To edit an equipment definition**

**1** In Toolspace, on the Survey tab, expand the Equipment Databases collection. Expand the named equipment database. Right-click the equipment definition. Click Edit.

**2** In the Equipment Properties dialog box, modify the properties as required. For more information about the equipment properties, see New Equipment/Equipment Properties Dialog Box (page 2384).

**To copy an equipment definition**

**1** In Toolspace, on the Survey tab, expand the Equipment Databases collection. Expand the named equipment database. Right-click the equipment definition. Click Copy.

**2** In the Equipment Properties dialog box, enter a name for the new definition.

**To delete an equipment definition**

**1** In Toolspace, on the Survey tab, expand the Equipment Databases collection.

**2** Expand the named equipment database. Right-click the definition. Click Delete.

## Quick Reference

Toolspace

Survey tab: right-click Equipment Databases
or
Survey tab: Equipment Databases ➤ right-click <database-name>
or
Survey tab: Equipment Databases ➤ <database name> ➤ right-click <equipment-definition-name>

# Survey Figure Prefix Database

The survey figure prefix database contains information to determine the layer that a figure is drawn on, how a figure is stylized, and whether figures are created as breaklines and lot lines.

When figures are imported or created, they are matched based on their names and the prefix names. When a figure is created and its name is matched with a figure prefix, the figure prefix properties are assigned to the new figure, such as a layer name.

Figure prefixes are also used to determine linework connectivity when you use the Process Linework command. Any point that is coded with a name that matches a figure prefix is treated as an active feature. The Process Linework command begins a figure when it encounters a prefix match if there is not an active figure with the same name.

For information about specifying the default path and location for figure prefix database, see Setting the Figure Prefix Database (page 208).

**To create a figure prefix database and definition**

**1** In Toolspace, on the Survey tab, right-click the Figure Prefix Databases collection. Click New.

**2** In the New Figure Prefix Database dialog box, enter a name for the new figure prefix database.

**3** To create a figure prefix definition, right-click the figure prefix database. Click New.

**4** In the Figure Prefix Properties dialog box, enter a name for the figure prefix definition and modify other properties as required. For more information about the figure properties, see New Figure Prefix/Figure Prefix Properties Dialog Box (page 2394).

**To make a figure prefix database current**

■ In Toolspace, on the Survey tab, expand the Figure Prefix Databases collection. Right-click the figure prefix database. Click Make Current.

**To edit a figure prefix definition**

1 In Toolspace, on the Survey tab, expand the Figure Prefix Databases collection. Click the named figure prefix database.

2 In the Survey list view, modify the properties of the figure prefix definition as required. For more information about the figure prefix properties, see New Figure Prefix/Figure Prefix Properties Dialog Box (page 2394).

**To copy a figure prefix database**

1 In Toolspace, on the Survey tab, expand the Figure Prefix Databases collection. Right-click the named figure prefix database. Click Copy.

2 In the Figure Prefix Databases dialog box, enter a name for the new database.

**To delete a figure prefix definition**

1 In Toolspace, on the Survey tab, expand the Figure Prefix Databases collection.

2 Click the named figure prefix database.

3 Right-click the definition in the Survey list view. Click Delete. Click 🖫 to save the delete operation.

### Quick Reference

Toolspace

Survey tab: right-click Figure Prefix Databases
or
Survey tab: Figure Prefix Databases ➤ right-click <database-name>
or
Survey tab: Figure Prefix Databases ➤ <database name> ➤ right-click <equipment-definition-name> in list view.

# Survey Networks

A survey network is a series of interconnected lines that represent the observed instrument setups, or stations. It contains all associated known control points, known directions, setups, and observations.

**TIP** Use the Import Survey Data command to create a network and import data using the wizard interface. For more information, see Import Survey Data Wizard (page 239).

After you have imported or created data in the survey network you can insert the network into your AutoCAD Civil drawing as a network object. For information about survey objects, see Survey Objects (page 194).

The following illustration shows the network components:
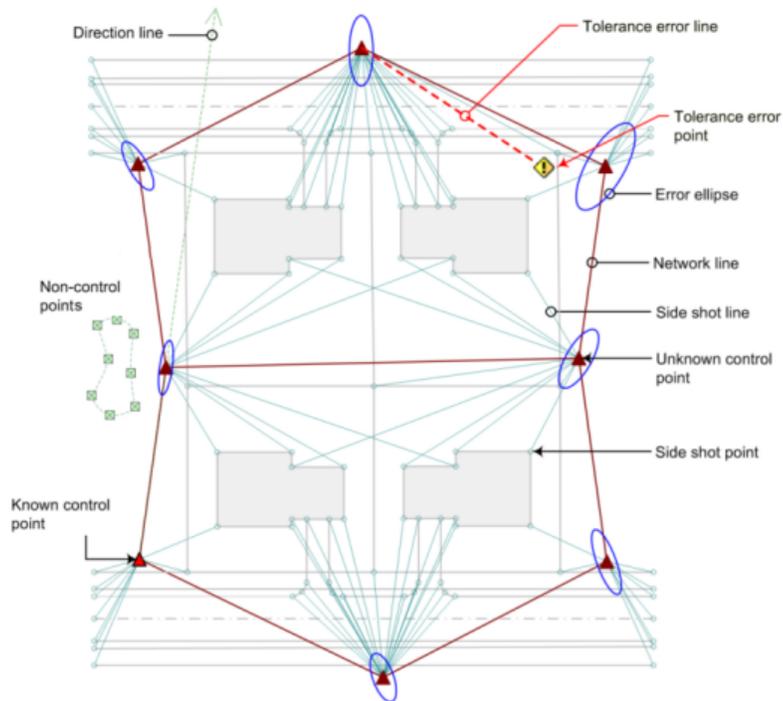
**Tips for working with Networks**

■ In a drawing you can hover your mouse over any network component to display a tooltip with component information.

■ In the drawing, select a network component and right-click for options to browse to the survey data on the Survey tab, edit, or update the data.

■ In Survey Toolspace, you can change the network display order using drag and drop functionality. When you drag and drop a network it is inserted after the network on to which it is dropped. Drag a network and drop it on the Networks collection to have it display first in the list.

■ You can drag and drop a Network from the Survey tab in Toolspace into the drawing.

■ Create a new network for each phase of a surveying project.

■ Create a new survey network to use the Survey Command Window to perform calculations involving points from other existing survey networks. Data that you input is captured in the batch file, where you can make edits and run the file again with the corrections, if necessary, without affecting the integrity of the other survey networks.

**To create a survey network**

■ In Toolspace, on the Survey tab, expand the survey project. Right-click the Networks collection. Click New.

**To insert a survey network into the drawing**

■ In Toolspace, on the Survey tab, expand the survey project, expand the Networks collection, right-click the named network, and click Insert Into Drawing.

■ In Toolspace, on the Survey tab, select the network and drag and drop the selection onto an open drawing in the AutoCAD Civil.

A survey network object is created in the drawing. Browse to the Survey collection in the Toolspace Prospector tab to view and modify the network object.

**To view survey network data in the drawing**

■ In the drawing, hover your cursor over a network object component to display a tooltip with relevant data.

**To browse to survey network data from the drawing to the Survey tab in Toolspace**

■ In the drawing, use CTRL + Select to select a network object sub-component, such as a sideshot point, and right-click to display commands for browsing to the survey data. The Survey tab is displayed (if hidden) and the survey database item is highlighted.

## Quick Reference

Toolspace

Survey tab: expand Projects ➤ <project name> ➤ right-click Networks ➤ New
or
Survey tab: expand Projects ➤ <project name> ➤ expand Networks ➤ right-click <named network> ➤ Insert Into Drawing

# Browsing to a Survey Network

Browse to a network in the Networks collection on the Survey tab in Toolspace.

The survey network is highlighted in the Networks collection on the Survey tab in Toolspace.

**To browse to a survey network**

■ Click Survey tab ➤ Modify panel ➤ Browse To Survey Data ➤ ▼ ➤ Browse To Network ⬚ .

■ Click Survey menu ➤ Browse ➤ Browse To Network. Double click a network name on the Survey tab in Toolspace to display the network in the drawing, or click a network in the drawing to highlight the network on the Survey tab.

## Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ Browse To Survey Data ➤ ▼ ➤ Browse To Network ⬚

Menu

Survey menu ➤ Browse ➤ Browse To Network

Command Line

BrowseToSvNetwork

# Survey Settings

# 9

Before using the AutoCAD Civil survey features, review and modify the settings that define survey user defaults, drawing settings, database settings, and equipment settings.

## User Settings

Survey user settings are specific to a Windows user login account and affect only the survey features, not project or drawing data.

Specify the default settings for importing and exporting survey data and the display of interactive graphic components.

**Tips for working with Settings**

- Default Survey Database settings can be set up and can then be used each time you create a new survey database.

- Survey database settings can be exported to file with a *.*sdb_set* extension.

- Survey User Settings can be exported to a *.*usr_set* file.

- You can create a .sdb_set for imperial or metric survey database settings, or for use with for specific coordinate zone.

- Have everyone in your organization use the same path for the Survey User Settings.

## Setting the Survey Database Defaults

Specify the settings for survey database features.

**To specify the survey database defaults**

1 Click Survey tab ➤ Modify panel ➤ ⚊ ➤ Edit User Settings 🖼 .

2 In the Survey User Settings dialog box, expand the Survey Database Defaults property group.

3 Under Survey database settings path, specify the path to where survey database settings files are located. This path should be in a common location within an organization.

**4** Under Survey database settings, select a survey database settings file from the drop list. All files with a *.sdb_set extension are listed. The settings in this file are used whenever a new survey database is created by AutoCAD Civil.

**5** Under, Extended properties definition path, specify the path to where Extended Properties definitions are located. This path should be a common location within an organization.

**6** Under Extended properties definition, specify an extended properties definition file from the list. All files in the extended properties definition path that has a *.sdx_def extension is listed.

For information about the setting equipment properties, see New Equipment/Equipment Properties Dialog Box (page 2384).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 🔻 ➤ Edit User Settings 🀫

Menu

Click Survey  menu ➤ Edit User Settings.

Toolspace

Survey tab: 🀫

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Setting the Current Equipment Database

Specify the current equipment, equipment database, and equipment database path.

The current equipment sets the values associated with a specific surveying instrument, such the standard deviations associated with the measuring capabilities for the equipment. This information is used in various calculations including Least Squares.

**To specify the current equipment database**

**1** Click Survey tab ➤ Modify panel ➤ 🔻 ➤ Edit User Settings 🀫.

**2** In the Survey User Settings dialog box, expand the Equipment Defaults property group.

**3** Under Equipment Database Path, enter the path for the survey equipment databases or click 🔍 to browse to the folder. This is the path where all new equipment databases are stored.

**4** Under Current Equipment Database, select the current database from the drop-down list.
The list of available databases is determined from the databases contained in the Equipment Databases collection in the Toolspace Survey tab.

**5** Under Current Equipment, select the current equipment from the drop-down list.

For information about the setting equipment properties, see New Equipment/Equipment Properties Dialog Box (page 2384).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼️

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: 🖼️

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Setting the Linework Processing Defaults

Specify the default path for the linework code set.

A linework code set interprets the syntax of the field codes that are entered into the data collector by the survey field crew.

**To specify the linework processing defaults**

1   Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼️.

2   In the Survey User Settings dialog box, expand the Linework Processing Defaults property group.

3   For Linework Code Sets Path, enter the path for the linework code or click 🔍 to browse to the folder. This is the path where all linework code set files are stored.

4   For Process Linework During Import, select the check box to process linework during the import. Linework can also be processed after the import is completed.

   **TIP** If you make adjustments to the associated network after importing data, update the linework using the Process Linework command when the adjustments are completed.

5   For the Current Linework Code Set, select a linework code set from the list.

6   For the Process Linework Sequence, select one of the following from the list:
   - By Import Order - processes points in the order which they are imported. Point names are always processed by import order.

   - By Point Number - processes points sequentially by point number (ascending order only).

For more information about linework processing settings, see Survey User Settings Dialog Box (page 2374).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼️

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: 📷

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Setting the Figure Prefix Database

Specify the current and default figure prefix database and database path.

Figure prefixes enable you to determine the layer that a figure is drawn on, how a figure is stylized, and whether figures are created as breaklines and lot lines. When figures are imported or created, they are matched based on their names and the prefix names. All figures with a specific prefix are assigned the properties of the prefix.

**To specify the figure prefix database**

1  Click Survey tab ➤ Modify panel ➤ 📷 ➤ Edit User Settings 📷.

2  In the Survey User Settings dialog box, expand the Figure Defaults property group.

3  For Figure Prefix Database Path, enter the path for the figure prefix databases or click 🔍 to browse to the folder. This is the path where all new figure prefix databases are stored.

4  For Current Figure Prefix Database, select the current database from the drop-down list.

The list of available databases is determined from the databases contained in the Figure Prefix Databases collection on the Toolspace Survey tab.

For more information about figure prefix settings, see Survey User Settings Dialog Box (page 2374).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 📷 ➤ Edit User Settings 📷

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: 📷

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Specifying External Editor Settings

Specify whether to use an external editor for displaying analysis input and output and for editing field book and batch files.

**To specify the external editor settings**

1   Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼.

2   In the Survey User Settings dialog box, expand the Miscellaneous property group.

3   For Use External Editor, select the check box to use an external editor.

> **NOTE** If you do not specify to use an external editor, the default editor, specified in the AutoCAD Options dialog box, is used.

4   For External Editor, enter the path and name for the editor or click 🔍 to browse to the editor.

## Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼

Menu

Click Survey  menu ➤ Edit User Settings

Toolspace

Survey tab: 🖼

Dialog Box

Survey User Settings Dialog Box (page 2374)

# Specifying Interactive Graphics Settings

Use Interactive Graphics settings to control the display of survey components during import and entry of survey data.

> **NOTE** The interactive graphics are temporary.

Examples of survey interactive graphics:
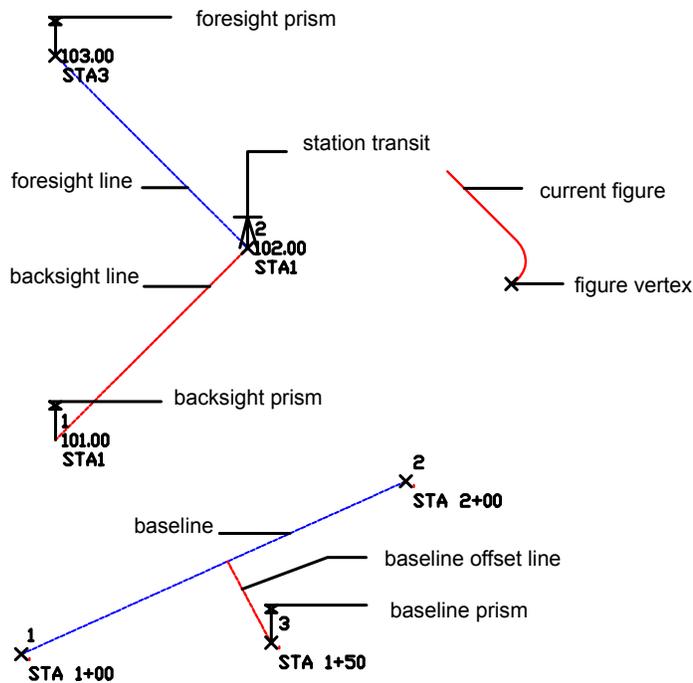
**To specify interactive graphics settings**

---

**NOTE** Using interactive graphics when you are importing a field book file significantly slows the import process.

---

1  Click Survey tab ➤ Modify panel ➤ 🔽 ➤ Edit User Settings 🛠️ .

2  In the Survey User Settings dialog box, expand the Interactive Graphics property group.

3  Select the check boxes for the interactive graphics that you want to display.
   For descriptions of the interactive graphic components, see Survey User Settings Dialog Box (page 2374).

4  Optionally, to change the colors for the components, click the color swatch to open the Select Color dialog box, from which you can select a color.

## Quick Reference

Ribbon

   Click Survey tab ➤ Modify panel ➤ 🔽 ➤ Edit User Settings 🛠️

Menu

   Click Survey  menu ➤ Edit User Settings.

Toolspace

   Survey tab: 🛠️

Dialog Box

   Survey User Settings Dialog Box (page 2374)

# Setting Import Defaults

Specify the Import Defaults properties for importing a field book or batch files into the selected project network.

**To specify import defaults**

1 Click Survey tab ➤ Modify panel ➤ <img> ➤ Edit User Settings <img>.

2 In the Survey User Settings dialog box, expand the Import Defaults property group.

3 Select the check boxes for the defaults that you want to enable.

For descriptions of the import defaults, see Survey User Settings Dialog Box (page 2374).

4 Optionally, for Default Figure Site property, enter a site name.

If the site does exists in the current drawing, it is created, or if it exists, it is used.

**NOTE** The site is used if the Lot Line property for the figure is true and the Site property for the figure is not specified.

## Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ <img> ➤ Edit User Settings <img>

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: <img>

Dialog Box

Survey User Settings Dialog Box (page 2374)

# Reporting Tolerance Errors

Display tolerance errors in the Event Viewer at the completion of survey data import or run the command from the network collection or an individual network on the Survey tab.

Tolerance errors are based on the values you specify in the Survey Database Settings (page 2378). At the completion of importing survey data into the survey database, tolerance errors (in the network) that exceed these values can be displayed as events in the Event Viewer. In the Survey User Settings dialog box (page 2374) - Import Defaults, specify Yes for Display Tolerance Errors In Event Viewer.

The Report Tolerance Error command can be run at anytime from the right-click shortcut menu on the Networks collection or an individual network.

You can override the default settings in the Import Survey LandXML (page 2420), Import Field Book (page 2406), and Batch File (page 2413) dialog boxes.

**To report tolerance errors at the completion of importing survey data**

■ In the Survey User Settings - Import Defaults, specify Yes for Display Tolerance Errors In Event Viewer. The event viewer is displayed at the completion of any import function if the data contains tolerance errors.

**To report tolerance errors**

■ On the Survey tab in Toolspace right-click the Networks collection or a <named> network ➤ Report Tolerance Errors.

## Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ right-click Networks collection or right-click <named> Network ➤ Report Tolerance Errors

## Tolerance Errors Display in the Event Viewer

Use the event viewer to review and edit tolerance errors.

If you have specified Display Tolerance Errors In Event Viewer in the Survey User Settings (page 2374) and if the network contains tolerance errors, the following items are displayed the Event Viewer at the completion of a survey import command.

**NOTE** You can also run the Report Tolerance Error command from the right-click shortcut menu on the Networks collection or an individual network.

| Column | Description |
| --- | --- |
| Tree tab | Displays the tolerance error and the associated network. |
| Type | Displays the tolerance error as a warning using the warning icon. |
| Date | Displays the system date of the event. |
| Time | Displays the system time of the event. |
| Source | Displays the name of the network that contains the tolerance error. |
| User | Displays the Windows user log in. |
| Description | Displays the Point ID and the tolerance error type. |
| More information | Click Zoom To to zoom to the tolerance error in the drawing. |
| Action | Click Browse To to display the Observations Editor. The observation with the tolerance error is highlighted. |

| Column | Description |
| --- | --- |
| | **NOTE** The Survey Database must be open to use the Browse To link. |

**See also:**

■ Event Viewer Vista (page 2455)

# Setting Export Defaults

Use the Export Defaults to specify the properties for exporting survey data to a field book file.

### Figure Export Conditions and Formats

When you select the Export Figures With Network property, the format of the figure data exported to the field book is dependent on the conditions of the survey data as shown in the following.

**NOTE** For more information about the syntax in the following section, see the Figure Commands (page 397).

■ If the figure vertex has a point number, then the figure vertex is defined by referencing the point in the field book file:
BEG <figure name>

PT <point ID>

■ Then the following lines are added to the field book file:
NE SS <point ID> <north> <east> <elevation> <description>

BEG <figure name>

PT <point ID>

■ Then the following line is added to the field book file defining the figure vertex:
FIG NE <north> <east>

■ If the figure segment is a curve, then the curve segment is defined using the XC ZD (BULB) command. The BULB parameter is used when the delta angle for the curve is greater than 180 degrees.

■ If both ends of the curve segment reference point IDs, then the following format is output:
XC ZD (AZ <point ID> <point ID>) (D <point ID> <point ID>)

■ Otherwise the following format is output:
XC ZD <Azimuth> <Distance>

### Point Export Conditions and Format

If you select the Export Point Identifiers property, for each figure vertex, the following formats are output to the field book file.

**NOTE** For more information about the syntax in the following section, see the Figure Commands (page 397).

■ If the figure vertex has a point ID, then the following lines are added to the field book file:
BEG <Figure name>

PT <point ID>

■ If the vertex does not reference a point ID, then the following lines are added to the field book file:
BEG <figure name>

FIG NE <north> <east>

- If Export Point Identifiers property is not selected, the following lines are added to the field book file:
  BEG <figure name>
  FIG NE <north> <east>

- If the figure segment is a curve, then the curve segment is defined using the XC ZD (BULB) command. The BULB parameter is used when the delta angle for the curve is greater than 180 degrees.

- If both ends of the curve segment reference point numbers (and if this check box is selected), then the following format is output:
  XC ZD (AZ <point ID> <point ID>) (D <point ID> <point ID>)

- Otherwise the following format is output:
  XC ZD <azimuth> <distance>

When the Export Point Data is enabled, for each figure vertex that references a point ID, the following format is output to the field book file prior to the section defining the figures:

NE SS <point ID> <north> <east> <elevation> <description>

**To specify export defaults**

1   Click Survey tab ➤ Modify panel ➤ ⌄ ➤ Edit User Settings 🖼.

2   In the Survey User Settings dialog box, expand the Export Defaults property group.

3   Select the check boxes for the defaults that you want to enable.
    For descriptions of the properties, see Survey User Settings Dialog Box (page 2374).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ⌄ ➤ Edit User Settings 🖼

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: 🖼

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Setting Display Preview Behavior

Use the Network Preview, Setup Preview, and Figure Preview settings to specify the preview properties when an item is selected in the Toolspace Survey tab.

**NOTE** The previews use the colors set by the Interactive Graphics settings. For information, see Specifying Interactive Graphics Settings (page 209)

**To specify display preview behavior**

1   Click Survey tab ➤ Modify panel ➤ ⌄ ➤ Edit User Settings 🖼.

**2** In the Survey User Settings dialog box, expand the Miscellaneous property group and edit the Preview Vertical Exaggeration setting.

**3** In the Survey User Settings dialog box, expand the Network Preview, Setup Preview, and Figure Preview property groups.

**4** Select the check boxes for the components that you want to display in the previews.

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼

Menu

Click Survey menu ➤ Edit User Settings.

Toolspace

Survey tab: 🖼

Dialog Box

Survey User Settings Dialog Box (page 2374)

## Importing and Exporting User Settings

Import and export existing user settings.

User settings can be exported to a user settings (.usr_*set*) file and subsequently imported.

**To export user settings**

**1** Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼 .

**2** In the Survey User Settings dialog box, click 🖼 .

**3** In the Save As dialog box, browse to the path where you want to save the settings file and enter a name.

**4** Click Save.

**To import user settings**

**1** Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit User Settings 🖼 .

**2** In the Survey User Settings dialog box, click 🖼 .

**3** In the Open dialog box, browse to the path where the settings file is located and select it.

**4** Click Open.
The Survey User Settings dialog box is populated with the settings from the file.

### Quick Reference

# Database Settings

Survey database settings are specific to the survey features of an AutoCAD Civil survey database.

By default, the survey database settings are stored in the *C:\Civil Projects\<database name>\Survey.sdb* file.

To edit survey database settings, the survey database must be open.

# Setting Measurement Units

Use the Units settings to specify project units including coordinate zone, distance, angle, direction, temperature, and pressure.

**NOTE** The survey database units are independent from the drawing units. If the survey database units and drawing units are different, the survey data is transformed when it is inserted into the drawing.

**To specify survey database measurement units**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database name, and click Edit Survey Database Settings.

2  In the Survey Database Settings dialog box, expand the Units property group.

3  For the Coordinate Zone property, click ⊡ and select the zone in the Select Coordinate Zone dialog box.

4  For the other unit settings, click the drop-down list and select the value for the unit type that you want to modify.

   For more information about the settings, see the Units section of the Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

## Setting Unit Display Precision

Use the Precision settings to specify the survey database display precision for units, including angle, distance, elevation, coordinates, and latitude and longitude.

The precision value specifies the number of digits to display to the right of the decimal.

**To specify display precision**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click the project name, and click Edit Survey Database Settings.

2  In the Survey Database Settings dialog box, expand the Precision property group.

3  Click the value for the unit, and enter the precision value.

   For more information about the settings, see the Precision section of the Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 　 ➤ Edit Database Settings 

Menu

Click Survey menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

## Setting Measurement Type Defaults

Use the Measurement Type Defaults to specify the default measurement types for the database when adding new survey data, including angle type, distance type, vertical type, and target type.

For example, the defaults are used for adding new observations in the Observations Editor.

**To specify the measurement type defaults**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database name, and click Edit Survey Database Settings.

2  In the Survey Database Settings dialog box, expand the Measurement Type Defaults property group.

3  Click the drop-down list and select the value for the measurement type that you want to modify.

For more information about the measurement type defaults, see Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit Database Settings 🗝

Menu

Click Survey  menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

## Setting Measurement Corrections

Use the Measurement Corrections settings to specify measurement corrections to apply to survey observations.

**To specify measurement corrections**

1   In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database name, and click Edit Survey Database Settings.

2   In the Survey Database Settings dialog box, expand the Measurement Corrections property group.

3   Select the check boxes for the corrections that you want to enable.

   For more information about the settings, see the Measurement Corrections section of the Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit Database Settings 🗝

Menu

Click Survey menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Survey Command Window

Equipment Correction Commands (page 400).

Dialog Box

Survey Database Settings Dialog Box (page 2378)

# Measurement Correction Formulas

The survey corrections are calculated using the following formulas.

### Curvature and Refraction Formula

The following formula calculates the vertical angle (in radians):

$$VA = OVA - \left\{ \frac{(1-k) \times SD}{2 \times R} \right\}$$

where:

- OVA: Old vertical angle (in radians)
- VA: Vertical angle (in radians)
- SD: Slope distance
- R: Spheroid radius (if a coordinate system zone is set, this value is obtained from the zone)
- k: Coefficient of refraction

### Atmospheric Conditions Formula

The following formula determines the slope distance:

$$SD = OSD + OSD \times \left[ RI - \left( \frac{C \times P}{273.2 + t} \right) \right] \times 0.000001$$

where:

- OSD: Old slope distance
- SD: Slope distance
- C: Constant for the EDM carrier (for example, Sokkia Lietz 0.86)
- P: Atmospheric pressure in mm Hg
- RI: Group refractive index for EDM carrier (for example, Sokkia Lietz 287.96)
- T: Dry air temperature in degrees Celsius

### Sea Level Formula

The following formula calculates the horizontal distance at sea level:

$$HD = OHD - \frac{(SE \times OHD)}{R}$$

where:

- OHD: Old horizontal distance
- HD: Horizontal distance
- SE: Station elevation
- R: Spheroid radius (if a coordinate system zone is set, this value is obtained from the zone)

**Collimation Formulas**

The following formulas determine the collimation:

Horizontal collimation:

■ For FACE 1: Horizontal Angle = Measured Horizontal Angle + Horizontal Collimation Error

■ For FACE 2: Horizontal Angle = Measured Horizontal Angle - Horizontal Collimation Error

Vertical collimation (all vertical angles are converted to zenith):

■ For FACE 1: Vertical Angle = Measured Vertical Angle + Vertical Collimation Error

■ For FACE 2: Vertical Angle = Measured Vertical Angle - Vertical Collimation Error

# Setting Traverse Analysis Defaults

Use the Traverse Analysis Defaults to specify the database setting defaults for performing a traverse analysis on survey data.

**To specify traverse analysis defaults**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the project name, and click Edit Survey Database Settings.

2  In the Survey Database Settings dialog box, expand the Traverse Analysis Defaults property group.

3  Edit the defaults as required.

For more information about the settings, see the Traverse Analysis section of the Survey Database Settings Dialog Box (page 2378).

## Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 🔻 ➤ Edit Database Settings 🖬

Menu

Click Survey menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

# Setting Least Squares Analysis Defaults

Use the Least Squares Analysis Defaults to specify the database setting defaults for performing a least square analysis on a network or a traverse.

**To specify least squares analysis defaults**

1  In Toolspace, click the Survey tab, expand the Survey Databases collection, right-click on the database name, and click Edit Survey Database Settings.

**2** In the Survey Database Settings dialog box, expand the Least Squares Analysis Defaults property group.

**3** Edit the defaults as required.

For more information about the settings, see the Least Squares Analysis section of the Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 🔽 ➤ Edit Database Settings 🗄

Menu

Click Survey menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

## Setting Survey Command Window Options

Use the Survey Command Window options to specify the settings that control interaction with the Survey Command Window.

For information about using the Survey Command Window, see Survey Command Window (page 2419).

**To specify survey command defaults**

**1** In Toolspace, click the Survey tab, expand the Survey Databases collection.

**2** Right-click the database name, and click Edit Survey Database Settings.

**3** In the Survey Database Settings dialog box, expand the Survey Command Window property group.

**4** Edit the settings as required.

For more information about the settings, see the Survey Commands section of the Survey Database Settings Dialog Box (page 2378).

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ 🔽 ➤ Edit Database Settings 🗄

Menu

Click Survey menu ➤ Edit Survey Database Settings.

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

Survey Database Settings Dialog Box (page 2378)

# Setting Error Tolerance

Use Error Tolerance to specify the maximum difference distance, angle, elevation, and coordinates in point observations.

If the maximum difference is greater than the specified value, the Event Viewer displays the errors. For more information, see Reporting Tolerance Errors (page 211).

**To specify error tolerance settings**

1   In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database, and click Edit Survey Database Settings.

2   In the Survey Database Settings dialog box, expand the Error Tolerance property group.

3   Edit the settings as required.

    For more information about the settings, see the Error Tolerance section of the Survey Database Settings Dialog Box (page 2378).

## Quick Reference

Ribbon

  Click Survey tab ➤ Modify panel ➤ ▼ ➤ Edit Database Settings 🖳

Menu

  Click Survey  menu ➤ Edit Survey Database Settings.

Toolspace

  Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Dialog Box

  Survey Database Settings Dialog Box (page 2378)

# Setting Extended Properties

Use Extended Properties to specify if new extended property definitions are to be automatically created during a Survey LandXML import, or to display warnings in the Event Viewer if required extended properties have not been assigned during an export of Survey LandXML data.

**To specify extended properties settings**

1   In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database, and click Edit Survey Database Settings.

2   In the Survey Database Settings dialog box, expand the Extended Properties property group.

3   Specify Yes to create new definitions automatically when you are importing a LandXML file.

4   Specify Yes to display warnings for required properties that are missing when you export a LandXML file.

### Quick Reference

## Importing and Exporting Survey Database Settings

Import or export existing survey database settings.

Database settings can be exported to a database settings (*.set*) file and subsequently imported.

**To export survey database settings**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click on the database name, and click Edit Survey Database Settings.

2  In the Survey Database Settings dialog box, click 🗊 .

3  In the Save As dialog box, browse to the path where you want to save the settings file and enter a name.

4  Click Save.

**To import survey database settings**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection

2  Right-click the database name, and click Edit Survey Database Settings.

3  In the Survey Database Settings dialog box, click 🗊 .

4  In the Open dialog box, browse to the path where the settings file is located and select it.

5  Click Save.
   The Survey Database Settings dialog box is populated with the settings from the file.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ right-click <database name> ➤ Edit Survey Database Settings ➤ 
or

Survey tab: Survey Databases ➤ right-click <database name> ➤ Edit Survey Database Settings ➤ 

# Equipment Properties

The survey equipment properties specify the values associated with a specific surveying instrument, such as the standard deviations associated with the measuring capabilities for the equipment. This information is used in the Least Squares calculations.

The equipment property groups are:

■ Miscellaneous. Specifies the name and description of the equipment that is displayed in the equipment database.

■ Angle. Determines how an instrument measures angles.

■ Units. Determines the unit types for angle and distance.

■ Electronic Distance Meter (EDM). Determines the Electronic Distance Measuring settings.

■ Prism. Sets the Prism accuracy, offset, and constant.

■ Standard deviations. Specifies the accuracy of the surveying equipment. These values are used in calculating the standard errors that are displayed in the Least Squares input file.

---

**NOTE** For detailed descriptions of the equipment properties, see New Equipment/Equipment Properties Dialog Box (page 2384).

---

**To set the equipment properties**

1 In Toolspace, on the Survey tab, expand the Equipment Databases collection, and expand the database name.

2 Right-click on the equipment name and click Properties.

---

**NOTE** For information about creating equipment databases, see Survey Equipment Database (page 199).

---

3 Edit the properties as required.

For more information about the settings, see the New Equipment/Equipment Properties Dialog Box (page 2384).

### Quick Reference

Toolspace

Survey tab: Equipment Databases ➤ <equipment database name> ➤ right-click <equipment name> ➤ Properties

Dialog Box

New Equipment/Equipment Properties Dialog Box (page 2384)

# Survey Drawing Settings

You can use survey drawing settings to specify the default behavior for drawing-related survey commands.

Drawing settings are handled in a standard way throughout AutoCAD Civil. You access drawing settings using the Toolspace Settings tab. You can control these settings at three levels: the drawing level, the object collection (feature) level, and the command level. It is important to understand how the different levels of settings work together. For more information, see Understanding Settings (page 61).

Use the Survey collection in the Settings tab to establish defaults for survey-related commands. You can change either survey-specific settings at this level or override the drawing ambient settings.

**NOTE** Overrides to the drawing ambient settings at the Survey collection level affect only the specified level. The drawing level settings are not changed.

The topics in this section describe only those settings that affect survey-related commands, and do not cover the drawing ambient settings that you can change at the survey collection level, even though those settings are displayed in the Survey Settings dialog box. For more information, see Specifying Ambient Settings (page 68).

**To change the survey feature settings**

1  In Toolspace, on the Settings tab, right-click Survey ➤ Edit Feature Settings.

2  In the Edit Feature Settings dialog box (page 2373), make changes to the existing settings.

3  Click OK to save the changed settings in the drawing.

## Quick Reference

Toolspace

Settings tab: right-click Survey ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings - Survey Dialog Box (page 2373)

# Survey Field to Finish

# 10

Refer to this section for a description of the commands you can use to import and process survey data in AutoCAD Civil.

Field to Finish Field to Finish (page 2474) is a general term used to describe the surveying process that determines the connectivity and symbology of points surveyed in the field that match the feature name and field code field code (page 2474) that is defined in a AutoCAD Civil linework code set linework code set (page 2477), and the description keys that are defined for the current drawing. A field code is contained within the description of a survey point, and typically contains both the name of the point feature or line feature, and a special line segment or curve segment code to indicate connectivity between survey points of the same feature. During the processing of linework, the linework code set interprets the syntax of the field code within a survey point description. When you insert points into the drawing from the survey database, the display of the survey points is determined by a raw description match with the description keys in the drawing.

For more information, see Description Keys (page 537).

## Preparing to Import Survey Data

To successfully bring data from the field into AutoCAD Civil, it is important to understand the interaction between field codes, the linework code set, description keys, and the figure prefix database.

Perform the following setup tasks before importing survey data:

- Establish consistent point coding and linework connectivity. For more information, see Field Codes, Figure Prefixes, and Description Keys (page 228).

- Create a Linework Code Set based on the point coding and linework connectivity. For more information, see Linework Code Sets (page 230).

- Define the Survey User Settings. For more information, see User Settings (page 205).

- Define the Survey Database Settings. For more information, see Database Settings (page 216).

- Create a Survey Equipment Database. For more information, see Survey Equipment Database (page 199).

- Create a Figure Prefix Database and figure prefixes. For more information, see Survey Figure Prefix Database (page 200).

- Create Description Key Sets and description keys. For more information, see Creating Description Keys (page 550).

- Import survey data based on a drawing template (*.dwt*) that contains established styles and layers. For more information, see Drawing Templates (page 109).

Perform the following tasks after importing survey data:

■ View data in an import event and re-import the data if necessary. For more information, see Import Events (page 249).

■ Edit survey point properties and reprocess linework. For more information, see Editing Survey Point Properties (page 251) and Processing Survey Linework (page 251).

■ Create a traverse and perform traverse analysis. For more information, see Survey Analysis and Output (page 361).

■ Use the Survey Command Window and survey command language to batch process and record coordinate geometry operations. For more information, see Survey Command Reference (page 391).

## Field Codes, Figure Prefixes, and Description Keys

Surveyors locate existing features on the ground using systems, such as a total station or a GPS unit, that are configured with data collectors. For each surveyed point, the surveyor assigns a field code that describes the point feature or line feature and that is then saved in the data collector. Line features contain a line command code that indicates whether the line is a beginning, a continuation, an end, a curve segment, or a line segment. In AutoCAD Civil, the syntax of the field code corresponds to a previously defined linework code set. The correspondence between the field codes and the linework code set allows for the following:

■ Automatic assignment of COGO point properties such as Layer, Symbol, and Label when inserted into the drawing

■ Automatic assignment of line feature properties, such as Layer, Color, Linetype, and Lineweight when inserted into the drawing

■ Line connectivity between the COGO points

An example of a field code is EP1 B, where:

■ EP = the standard abbreviation that a company uses to represent an edge of pavement

■ 1 = the first edge of pavement that is located

■ B = the code in the linework code set to begin a figure

**NOTE** NOTE: When you import survey data, you can omit the Begin code if the feature name matches a figure prefix that is defined in the current figure prefix database. If the feature name does not match a figure prefix, you must specify a Begin code.

Field codes are associated with both the figure prefix database and the description keys in the current drawing. If EP has been defined in the figure prefix database, then EP1 matches EP and is assigned the properties of the EP figure prefix, such as layer, figure style, breakline. If the survey point with the description EP1 B is placed in a drawing that has a description key of EP*, then EP1 B matches description key EP* and is assigned the point properties defined in the EP* description key, such layer, point style, point label style.

The following table lists examples of features coded in the field and the field code assigned to the feature, and the description keys and Figure prefixes created in AutoCAD Civil to support the field codes:

| Feature | Field Code | Description Key | Figure Prefix |
| --- | --- | --- | --- |
| Edge of pavement | EP<ID> | EP* | EP |
| Utility Pole | UP<ID> | UP* | |

| Feature | Field Code | Description Key | Figure Prefix |
|---|---|---|---|
| Fence Line | FCE<ID> | FCE* | FCE |
| Centerline | CL<ID> | CL* | CL |
| Ditch Line | DL<ID> | DL* | DL |
| Edge of Gravel | EG<ID> | EG* | EG |
| Sewer Manhole | SMH<ID> | SMH* | |
| Storm Manhole | DMH<ID> | DMH<ID> | |

The following illustration shows an example of feature definitions:
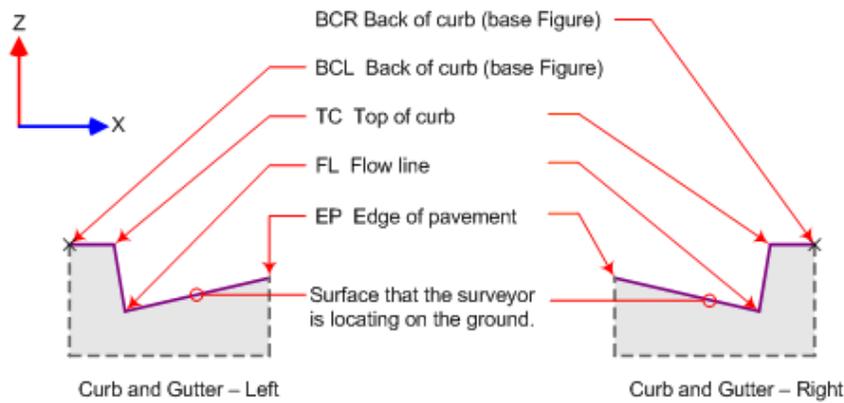


The following illustration shows these feature codes:

- 1: BC (back of curb)
- 2: TC (top of curve)
- 3: FL (flow line)
- 4: EP (edge of pavement)
- 5: CL (centerline of existing roadway)
- 6: Point located in the field by a surveyor
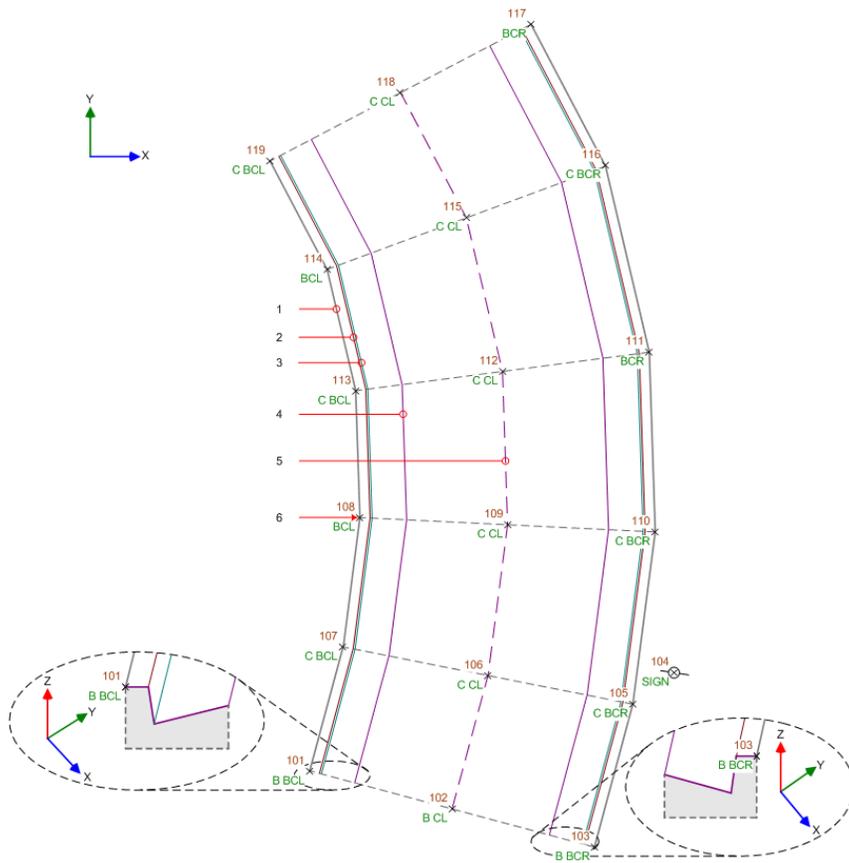
## Linework Code Sets

A linework code set interprets the syntax of the field codes that are entered into the data collector by the survey field crew. Linework Code sets are displayed on the Toolspace Survey tab. Specify the default linework code set in the Survey User Settings. An import event  import event (page 2476) stores the linework code set that was last used to process linework for that event.

**To create a new linework code set**

1   On the Survey tab in Toolspace, right-click the Linework Code Sets collection ➤ New.

2   In the New Linework Code Set dialog box, enter a unique file name.

3   In the Edit Linework Code Set dialog box, specify values for the following:

- Information

- Coding Methods

- Special Codes

- Line Segment Codes

- Curve Segment Codes

For more information, see Examples - Linework Codes (page 231).

4   Click OK.

For more information, see Edit Linework Code Set Dialog Box (page 2433).

**To copy or edit an existing linework code set**

1 On the Survey tab in Toolspace, expand the Linework Code Sets collection.

2 Right-click the existing linework code set and click Copy or Edit.

3 In the Edit Linework Code Set dialog box, enter a unique file name.

  By default, the Edit Linework Code Set dialog box displays the values of the copied file.

4 Edit the values and click OK.

### Quick Reference

Toolspace

Survey tab: right-click Linework Code Sets ➤ New, Edit, or Copy

# Examples - Linework Codes

The following tables provide examples of linework code set properties.

## Coding Methods

**WARNING** It is recommended that you use <space> as the Field/Code Delimiter Property Value. Using decimal characters as the Field/Code Delimiter Property Value may cause certain codes to fail. For example the (decimal). character is valid for a right-turn value and the (minus sign) - character is valid for a negative right-turn value.

| Property | Example | Description |
| --- | --- | --- |
| Feature/Code Delimiter: code that indicates the separation of the feature name from the code. | **EP1 B SW1 B** | A " " (space) is the recommended delimiter. |
| Field Code Escape: code that indicates that anything entered after the escape indicator is a comment. | **EP1 SW1 B/Start of sidewalk** | A / (forward slash) is the recommended escape. **Start of sidewalk** is a comment. You can enter and exit comment mode using multiple slashes. In the following example, /parking space line/is a comment and ignored. EP1 B/parking space line/LN CPN211. |

## Special Codes

| Property | Example | Description |
| --- | --- | --- |
| Begin: code that indicates the start of a new figure at the specified point. | **EP1 B SW1 B** or **B EP1 B SW1** | **EP1** and **SW1** are figure names. The letter **B** is the special code used to begin new figures named **EP1** and **SW1**. |

| Property | Example | Description |
|---|---|---|
| If a figure prefix exists that matches the figure name, you can omit the Begin code. This works best with unique figure names. | | |
| Continue: code that indicates a figure is continued from its last vertex. | EP1 C SW1 | The figure is continued with an explicit \<Continue code\>. **EP1** is a figure name, **C** continues the active figure named **EP1** If the field code does not contain an explicit \<Continue\> code and the figure name within the field code matches an active figure, the figure is continued. |
| End: code that indicates the figure is terminated. | EP1 E SW1 B | **EP1** is the figure name, " " (space) is the Field/Code Delimiter, **E** is the End code and continues an active figure with the name **EP1** to this point and then it is terminated (it is no longer an active figure). **SW1** is a figure name, " " (space) is the Field/Code Delimiter, and **B** is the Begin code and starts a new figure named **SW1**. In this example, if there is an existing figure named **SW1** it is terminated at its last vertex, and a new figure named **SW1** is started at this point. |
| Close: code that indicates that an active figure is continued from its last vertex to the current point and closes the figure with a line segment back to the starting vertex for that figure. | EP1 SW1 CLS | **EP1** is a figure name and if it matches an active figure it is continued to this point. **SW1** is a figure name and if it matches an active figure it is continued from its last vertex to this point, **CLS** is the Close code and a line segment is closed back to the starting vertex for the figure SW1. |
| Horizontal Offset: code used to specify a relative horizontal offset for creating a parallel figure starting at the current point, or transitioning to another offset if the active figure has a previous horizontal offset. | 101,500,490,100.01,BC1 B H-4 V.1 H.5 H.75 V-.7 H2.25 V-.35 ... 112,500,500,100.02,BC1 ... 123,500,510,100.03,BC1 ... 134,500,520,100.04,BC1 ... 145,500,530,100.05,BC1 ... 156,500,540,100.06,BC1 ... 167,500,550,100.07,BC1 SO ... 178,500,560,100.08,BC1 | Point 101: **BC1** is the line feature name, " " (Space) is the Feature/Code delimiter, **B** is the \<Begin\> code, **H** is the \<Horizontal offset\> code, -4 is the value for the first horizontal offset value, **V** is the \<Vertical offset\> code, .1 is the first vertical offset value, and so on for each of the remaining 3 offsets. Points 112, 123, 134, 145, and 156: BC1 figure with current offsets are continued. Point 167: **BC1** is continued, but the offset figures are stopped by the **SO** \<Stop offsets\> code. Point 178: **BC1** is continued. A value must follow the Horizontal offset code. A negative value offsets the active figure to the left and a positive value offsets the active figure to the right. Multiple offsets can be used, but if subsequent horizontal or vertical transitions are made, all previous horizontal offset or vertical offset codes must be specified for each offset. |

| Property | Example | Description |
|---|---|---|
| Vertical Offset: code used to specify a relative vertical offset starting at the current point. A value must follow the Vertical offset code. | (Shown in the following illustration) | A positive value is added to the elevation of the current vertex for the active figure and a negative value is subtracted from the current vertex for the active figure. If a Vertical offset is specified with no previous Horizontal offset, the Horizontal offset value is assumed to be zero. If a Horizontal offset is specified with no Vertical offset or no previous Vertical offset, the Vertical offset value is assumed to be zero. |
| | | **NOTE** If a previous horizontal or vertical offset was specified in the current point code, and if a subsequent horizontal or vertical offset is omitted within the current field code, the previous horizontal or vertical offset value is used. For example: |
| | | V1 H1 H2 = H0 V1 H1 V1 H2 V1<br>H1 H2 V1 H3 = H1 V0 H2 V1 H3 V1 |
| Stop Offsets: code used to specify the termination of all offsets on the active figure. Only the active figure can be continued. | (Shown in the following illustration) | |

The following illustration shows an example using the codes for Horizontal Offsets, Vertical Offsets, and Stop Offsets:

1,500,490,100.01,BC1 B H-4 V.1 H.5 H.75 V-.7 H2.25 V-.35

2,500,500,100.02,BC1

3,500,510,100.03,BC1

4,500,520,100.04,BC1

5,500,530,100.05,BC1

6,500,540,100.06,BC1

7,500,550,100.07,BC1 SO

8,500,560,100.08,BC1

## Line Segment Codes

| Property | Example | Description |
| --- | --- | --- |
| Recall Point: code used to connect the active figure with a segment from the last point, or a specified pointID. | **EP1 RPN**<br>**EP1 RPN101** | **EP1** is the figure name and continues an active figure if the name matches. **RPN** is the Recall Point code and if there is no point it connects from the previous point to the current point and inserts a segment before the current point. The code EP1 RPN101 connects from the current point to the indicated point. |
| Connect Point: code used to indicate the creation of a new figure (of the same feature) with a single line segment from the current point to the specified point ID. The specified point ID follows the Connect Point code. | **EP1 B CPN101** | In this example, a new figure is created at the current point, and also another new figure with a single line segment is drawn to point 101, called EP1.CPN101. The figure name can be modified using the Figure Properties command. |

The Connect Point code always creates a figure composed of a single line segment. The following illustration shows an example of the Connect Point code:

| Property | Example | Description |
| --- | --- | --- |
| Rectangle: code used to offset the segment coming into the current point by the specified number. | **BLD1 <Rectangle>40** | A positive number indicates an offset to the right, a negative number indicates an offset to the left, which is relative to the direction of the line segment coming in to the current point. If no number follows the Rectangle code, this code closes the figure by performing a perpendicular/perpendicular line intersection between the previous segment coming into the current point and the first segment of the figure. |
| | | **NOTE** A Rectangle code without a number requires at least three points previously located on the current figure. |

The following illustration shows the Rectangle code with a specified value of 40:



| Property | Example | Description |
| --- | --- | --- |
| Right Turn: code used with a line segment code to allow for additional vertices to be inserted perpendicular into the figure, or as extensions (straight offsets). | **BLD1 RT X10.1 5 -12.2 -5 -12.2** | Continues an active figure **BLD1** to the current point, extends the current segment **10.1** units, and then draws perpendicular segments for each value. Offset values, <Extend>, or <Rectangle> codes follow the <Right turn> code. Positive numbers indicate a jog to the right and negative numbers indicate a jog to the left. |
| | | **NOTE** Use <Extend><Value> to make an extension straight ahead by using a positive # or behind by using a negative #. |
| | | **NOTE** Use the <Rectangle> code to complete the <Right turn> code, closing back to the starting segment as a perpendicular/perpendicular line intersection from the current figure line segment. |

The following illustration shows examples of the Right Turn code and the Rectangle code:

| Property | Example | Description |
|---|---|---|



| Extend: code that is used to make an extension of a line segment ahead through the current point by using a positive value or a line segment that is short of the current point by using a negative value. | **BLD1 X15.5** | **BLD1** continues an active figure, X is the Extend code, and **15.5** is the value that the figure line segment is extended through the current point. |

The following illustrations show examples of the Extend code, Right Turn code, and Rectangle code:



# Curve Segment Codes

| Property | Example | Description |
|---|---|---|
| Begin Curve: code used to specify that the current point is the beginning of a single curve (arc) segment or multiple curve segments. | **EP1 BC** | **EP1** is the active figure name and is continued to the current point, " " (space) is the Feature/Code Delimiter, **BC** indicates that the current point is the beginning of curve segments.<br>The current point with this code is the first point on the curve. The next point with the same figure name is considered a point on the curve, and the third point with the same figure name is the curve endpoint if no <End curve> code is used to correspond with the <Begin curve> code. |
| End Curve: code used with the <Begin curve> code to define curve segments with more than three points. Points on a curve do not have to be located sequentially. | **101 … EP1 BC**<br>**112 … EP1**<br>**123 … EP1**<br>**134 … EP1**<br>**145 … EP1 EC** | **EP1** is the active figure name and is continued to the current point, " " (space) is the Feature/Code Delimiter, **EC** indicates that the current point is the end of curve segments. |

| Property | Example | Description |
|---|---|---|
| | | Starting at the point with the <Begin curve> code, the linework processor will look for an <End curve> code. If the <End curve> code is found, all the points between the <Begin curve> code and <End curve> code are used to fit arc segments of the figure. The curves pass through each of the points (similar to the PEDIT/Fit [curve] option). If no <End curve> is found following a <Begin curve> code, then a three-point arc is applied to the figure. |
| Circle: code that indicates the beginning of the circle feature. This code stops the linework on the previous point if it is a two or three-point circle.<br>A one point circle with a radius value is similar to the connect point command. It escapes out and creates a new circle feature using the point as the center point and the radius. The two and three-point circles behave like begin commands to create a new figure and end an active feature with the same name. | **POOL CIR5.0** | **POOL** is the figure name, " " (space) is the Feature/Code Delimiter, **CIR** is the <Circle> code, and creates a new circular figure where the current point is the radius point, and **5.0** is the circle radius value. The elevation of the figure is the elevation of the current point. This example uses only the current point as the center point, with the <Circle> code followed immediately by the radius.<br>Method 1 uses only the current point as the center point, with the <Circle> code followed immediately by the radius.<br>Method 2 uses two consecutive points to define the circular figure. The first point defines the center location and the second point defines the elevation and the radius.<br>Method 3 uses three points to define a circular figure. The elevation of all three points is used and continues the closure with an arc instead of a line. |
| Point On Curve: code to indicate that the current point resides on a curve segment. | **EP1 OC** | **EP1** matches the name of an active figure, " " (space) is the Field/Code Delimiter, and **OC** is the Point On Curve code. The figure is continued and the point is evaluated as a point residing on a curve. If there is no match to an active figure, **EP1 OC** may be a point coding error. |

| Property | Example | Description |
|---|---|---|

Curve Fit - No curve: In the following illustration, each survey point has the same EP1 figure name and are line points (no <Point on curve> code indicating a curve point, or <Begin curve> code). The resulting figure is a series of connected line segments.
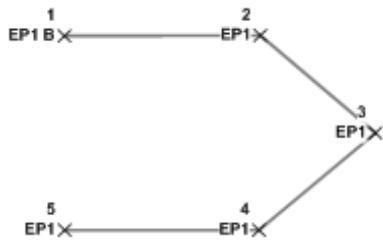


Curve Fit - One point on curve: In the following illustration each point has the same EP1 figure name, and one survey point is a coded as a curve point (OC <Point on curve> code). The resulting figure will have a calculated point of curvature (PC) and point of tangency (PT), and the curve will pass through the point.
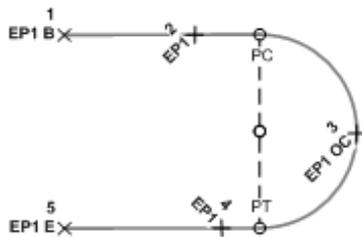


Curve Fit - Three-point curve: In the following illustration, each point has the same EP1 figure name, and there is three consecutive curve points (OC <Point on curve> code, or BC <Begin curve> code at the beginning of the curve). The resulting figure will have a begin curve (BC) at the first EP1 point, an end curve (EC) at the third EP1 point, and the curve will pass through the second EP1 point.
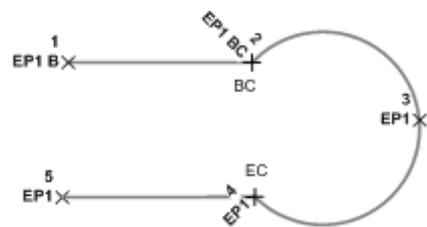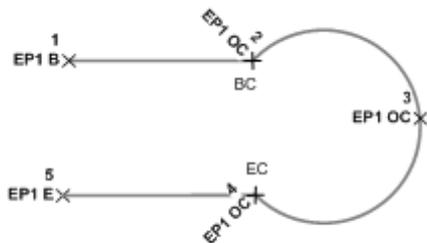




Curve Fit - Multi-point curves: In the following illustration, each point has the same CL1 figure name, and there are more than three consecutive points with an OC <Point on curve> code, or the first point on the curve segments has a BC <Begin curve> code and the ending point of the curve segments has an EC <End curve> code. The resulting figure will have a beginning of curve (BC) at the first CL1 point, and an end of curve (EC)

| Property | Example | Description |
|---|---|---|

at the last CL1 point. Compound and reverse curves will pass through each point between the BC figure vertex and the EC figure vertex.
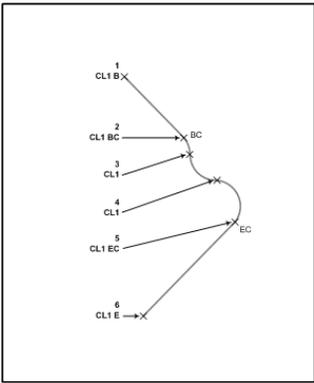


# Importing Survey Data

Import data using the wizard or import commands in the survey network.

## Import Survey Data Wizard

Use the wizard to easily import survey data.

In the Import Survey Data wizard you have options for importing field book files, point files, points from the drawing, or Survey LandXML files. Importing data using this method creates an Import Event. For more information, see Import Events (page 249).

**To import survey data using the wizard**

1  Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data .

2  On the Specify Database page, select an existing database or click Create New Survey Database.

3  On the Specify Data Source page, select one of the following options:
   ■  Field Book File

   ■  LandXML file

   ■  Point File

   ■  Points From Drawing

4  On the Specify Network page, specify a network or click Create New Network.

5  On the Import Options page, specify the import settings.

6  Click Finish.

An import event is created in the Import Events collection for the survey database.

**Quick Reference**

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data 

Toolspace

Survey tab: expand Survey Databases ➤ expand <named> database ➤ right-click Import Events ➤ Import Survey Data

Command Line

ImportSurveyData

# Field Book Files

Import field book files to enter data into the survey database and drawings.

Field book files (*.fbk*) are text files containing observed point data and figure definitions, which you can import into a drawing and database. Use the Survey Data Collection Link command to transfer and convert raw survey data to a field book file (*.fbk*) that you can import into AutoCAD Civil. For more information, see Survey Data Collection Link (page 240).

## Survey Data Collection Link

Use the Survey Data Collection Link command to transfer data from data collectors and convert the raw data files and coordinate data files into field book (*.fbk*) files.

Then you can import the *.fbk* files using the Import Survey Data Wizard or the import command in the survey network.

**NOTE** If you are using Windows Vista, you cannot display the Survey Data Collection Link Help files (Link32.hlp and Survey Attribute Manager.hlp) by default. To display the Survey Data Collection Link Help files in Windows Vista, download and install the Windows Help program (WinHlp32.exe) from the Microsoft Download Center.

## Transferring and Converting Data Using Survey Data Collection Link

Transfer and convert raw data files and coordinate data files into field book (*.fbk*) files that you can import into AutoCAD Civil.

**To transfer data**

1 Click Home tab ➤ Create Ground Data panel ➤  ➤ Survey Data Collection Link  to display Survey Link DC.

2 Click Transfer menu ➤ Send/Receive.

**To convert data files**

1 Click Home tab ➤ Create Ground Data panel ➤  ➤ Survey Data Collection Link  to display Survey Link DC.

2 Click Conversions menu ➤ Convert File Format.

**NOTE** For more information, use the Survey Link DC Help.

## Quick Reference

Ribbon

  Click Home tab ➤ Create Ground Data panel ➤ ⌷ ➤ Survey Data Collection Link 🖳

Menu

  Click Survey menu ➤ Survey Data Collection Link

Command Line

  StartSurveyLink

## Supported Input Types

The Survey Data Collection Link Extension supports the following input types.

| | |
|---|---|
| AutoCAD DXF | User Defined ASCII |
| AutoCogo | WILDSoft |
| ASCII (N,E,Z,Note) | Surv CAD |
| CLM | ASCII 2 |
| Autodesk-Softdesk | PacSoft Coordinates |
| ASCII | Maptech |
| Sierra Cybernetics | MTI |
| Drafix | Lewis & Lewis |
| TDS Coordinates | C & G |
| TDS Raw Data (*.rw5*) | TDS Raw Data (.raw) |
| Sokkia SDR Numeric 4 | Sokkia SDR Alpha 14 |
| Topcon FC-4 | Leica GRE3 |
| SMI | Nikon AP800,700,DTM500,400,300,DR48 |
| Topcon GTS210/220/310 Raw Data | Topcon DL-100 Digital Level |
| Leica GSI Raw Data | |

## Importing a Field Book File

Enter data into your drawing and survey database by importing an existing field book file.

**NOTE** In the Import Settings, clear the Process Linework check box when you are importing . *fbk* files created from the Survey Data Collection Link application because Linework Codes Set definitions may not be recognized. However, the figure commands created using the Survey Data Collection Link are recognized.

**TIP** You can also use the Import Survey Data wizard to import field book files. For more information, see Import Survey Data Wizard (page 239).

**To import a field book file from a network**

1  In Toolspace, on the Survey tab, expand the Networks collection and right-click the network to which you want to import the field book.

2  Click Import ➤ Import Field Book.

3  In the Field Book Filename (.FBK) dialog box browse to the field book that you want to import. Click Open.

4  In the Import Field Book dialog box, modify settings as required. For information about the settings, see Import / Re-import Field Book Dialog Box (page 2406).

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data 🗊

Toolspace

Survey tab: Survey Databases ➤ <database-name> ➤ Networks ➤ right-click <network-name> ➤ Import ➤  Import Field Book

# Editing a Field Book File

Use AutoCAD Civil to view and edit an existing field book file.

Load a field book file into a text editor so that you can view or edit it.

**NOTE** Notepad is the text editor unless you specify another editor in the External Editor in the Survey User Settings dialog box. For more information, see Specifying External Editor Settings (page 208).

**WARNING** To work correctly, a field book file must be in an ASCII format.

**To edit a field book file**

1  In Toolspace, on the Survey tab, expand the Networks collection and right-click on a network name.

2  Click Edit Field Book.

3  In the Field Book Filename (.FBK) dialog box, browse to the field book that you want to edit. Click Open.

4  The field book contents are displayed in the text editor.

5  Make any necessary changes and save the file.

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database-name> ➤ Networks ➤ right-click <network-name> ➤ Edit Field Book

## Exporting a Field Book File

Export survey data from an individual network, individual figure, or collection of figures to a field book file.

**To export a field book file**

**NOTE** A field book can be exported from networks and figures.

1. In Toolspace, on the Survey tab, do one of the following:
   - Expand the Networks collection and right-click the network from which you want to export data to a field book.
   - Right-click the Figures collection that you want to export to a field book.
   - Expand the Figures collection and right-click a figure name to export it to a field book.

2. Click Export Field Book.

3. In the Save As dialog box, browse to the location where you want to save the field book and enter a file name. Click Save.

4. If you are exporting a network to a field book, in the Export Field Book dialog box, modify the settings as required. For information about the settings, see Export Field Book Dialog Box (page 2413).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database-name> ➤ Networks ➤ right-click <network-name> ➤ Export Field Book
or
Survey tab: Survey Databases ➤ <database-name> ➤ Figures ➤ right-click <figure-name> ➤ Export Field Book

## Survey LandXML Data

You can import and export survey LandXML data directly into or from the survey database.

Use the AutoCAD Civil Import Survey LandXML command to import LandXML survey data directly into the survey database. You can subsequently insert the data into the drawing, using the Insert Into Drawing commands.

The following table illustrates the results of importing LandXML data using the Import Survey LandXML command.

| LandXML Data | Results Using Import Survey LandXML |
|---|---|
| Parcels | Figures in the survey database |
| PlanFeatures | Figures in the survey database |
| COGO points, survey monuments | Survey points in the survey database |
| Reduced observations | Setups and Observations, control points, or non-control points in the survey database |

| LandXML Data | Results Using Import Survey LandXML |
| --- | --- |
| Raw observations | Setups and Observations in the survey database |

**NOTE** You can import reduced survey data directly into a AutoCAD Civil drawing. For more information about the different types of LandXML import functionality for survey data, see Importing Survey XML Data.

You can also export LandXML data from the survey database.

You specify both the LandXML import settings and data selection using the Import Survey LandXML command.

The LandXML import and export functionality is based on the LandXML schema. For more information about the LandXML schema, go to *http://www.landxml.org*.

## Importing Survey LandXML Data

Enter data into the survey database and the drawing by importing an existing LandXML.

**TIP** You can also use the Import Survey Data wizard to import Survey LandXML files. For more information, see Import Survey Data Wizard (page 239).

**NOTE** You can only import a LandXML file into an open survey database. For information about creating and opening a survey database, see Survey Database (page 196).

**To import a LandXML file**

1  On the Survey tab in Toolspace, right-click a survey database ➤ Import Survey LandXML.

2  In the Select LandXML File dialog box, browse to the LandXML file that you want to import. Click Open.

3  In the Import Survey LandXML dialog box, modify the import settings and select the survey data that you want to import. For information about the import settings, see Import / Re-import Survey LandXML Data Dialog Box (page 2420).

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data 🗄

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Import Survey LandXML

## Exporting Survey Data to LandXML

Export survey data to a LandXML file using AutoCAD Civil.

**To export survey data to a LandXML file**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection and right-click a database name.

**2** Click Export Survey LandXML.

**3** In the Export Survey LandXML dialog box, modify the export settings and select the survey data that you want to export. For information about the settings, see Export Survey LandXML Data Dialog Box (page 2423).

### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ right-click <database-name> ➤ Export Survey LandXML

## Managing Survey Extended Properties

Use extended properties to import and export Survey LandXML attributes and to create, assign, and view user-defined attributes.

Using the extended properties command, you can do the following:

■ Import Survey LandXML data and attributes into the survey database.

■ View Survey LandXML attributes and modify user-defined attributes using the AutoCAD Civil Survey user interface.

■ Export Survey LandXML data (with attributes) to a specified LandXML file.

■ Insert Survey Points into a drawing with the Survey LandXML and user-defined attributes assigned.

■ Complete round-tripping of Survey LandXML and user-defined data using the Import Survey LandXML and Export Survey LandXML commands on the Survey tab in Toolspace.

## Extended Properties User Interface

By default there are no extended properties defined in the *sdx_def* file.

To display extended properties, you must specify an *sdx_def* file that contains definitions, or import a LandXML file with the Create New Definitions Automatically option selected in the Survey Database Settings.

For more information, see Survey Database Settings dialog box. (page 2378)

**To display the survey extended properties**

**1** In the Survey Database Defaults specify the path and the extended properties definition file.

**2** In Toolspace, on the Survey tab, expand the Survey Databases collection and right-click a database name ➤ Manage Extended Properties.

**3** In the Manage Extended Properties dialog box, select the check boxes for collections or individual properties.
For more information, see the Manage Extended Properties dialog box. (page 2425)

### Quick Reference

## Creating and Editing User-defined Properties

Use the options in the Manage Extended Properties dialog box to create or edit a user-defined property.

**To create or edit a user-defined extended property**

1 In Toolspace, on the Survey tab, expand the Survey Databases collection and right-click a database name.

2 Click Manage Extended Properties. In the Manage Extended Properties dialog box, collections are displayed for each of the <SurveyFeatureClasses>.

3 In the Manage Extended Properties dialog box, click ⫶▾ to expand the <SurveyFeatureClasses>in the tree. Each feature class has a collection for LandXML and User-defined.

4 Select a User-defined check box under a feature class, right-click ➤ New or Edit.

5 In the Edit Extended Property dialog box, enter the following:

■ XML Name

■ Display Name

■ Description

6 Select a type in the Property Field Type list. For more information on the display of property fields, see Property Field Type Table (page 246).

7 Select the Required Value check box to display a warning icon for properties that have not been assigned to an instance of a <SurveyFeatureClass>. The Event Viewer will also post a warning when you export the LandXML file using the Export Survey LandXML command.

8 Select the Display check box to have the extended property display on the object.

9 If you specified String as the Property Field Type, enter values for String.

### Quick Reference

## Property Field Types Table

Use this table to determine the display of the various Property Field types both in the Survey user interface and the display of extended properties that are assigned to Survey points in the drawing.

| Property Field Types | Stored in Survey Database As | Survey User-interface Display | DWG Formatting |
|---|---|---|---|
| Angle | Angle in radians | SDB Angle | Angle |

| Property Field Types | Stored in Survey Database As | Survey User-interface Display | DWG Formatting |
|---|---|---|---|
| Area | Square Units in Meters | SDB Square Distance | Area |
| Azimuth | Direction in Radians | SDB Direction | Direction |
| Boolean | Boolean | Boolean | Boolean |
| Coordinate | Distance in Meters | SDB Distance | Coordinate |
| Dimension | Distance in Meters | SDB Distance | Dimension |
| Direction | Direction in Radians | SDB Direction and Angle | Direction |
| Distance | Distance in Meters | SDB Distance | Distance |
| Double | Double | Double | Unitless |
| Elevation | Elevation in Meters | SDB Distance | Elevation |
| Grade | Double | Double | Grade |
| Slope | Double | Double | Slope |
| Integer | Integer | Integer | Unitless |
| Latitude | Angle in Radians | Signed Angle | Latitude |
| Longitude | Angle in Radians | Signed Angle | Longitude |
| Percent | Percent | Percent | Percent |
| Station | Distance in Meters | SDB Distance | Station |
| String | String | String | String |
| Volume | Cubic Meters | Cubic Yards if Foot or Cubic Meters if Meters | Volume |

## Extended Property Definition Files

You can edit default *.sdx_def* files or specify a pre-defined schema with specialized LandXML and User-defined properties contained in an existing *.sdx_def file.

By default, AutoCAD Civil Survey installs the LandXML - Standard *.sdx_def* and Sample *.sdx_def* files at the following location:

C:\ Documents and Settings\All Users\Application Data\Autodesk\C3D2010\enu\Survey.

For example, the LandXML - Standard *.sdx_def* file contains numerous SurveyFeatureClass items. You can open this file in a text editor and make any necessary edits to streamline this file to meet your needs. Use the Save As command and then save the file with a new name.

## Using Groups to Manage Survey Data

Use Network Groups, Figure Groups, and Survey Point Groups to organize and manage survey data.

Organize LandXML collections of elements, such as CgPoints or PlanFeature collections, using Network Groups, Figure Groups, and Survey Point Groups. These groups are located at the root collection within a Survey database item on the Survey tab in Toolspace.

**To create a new Network Group, Figure Group, and Survey Point Group**

1  In Toolspace, on the Survey tab, expand the Survey Databases collection and expand a database name.

2  Right-click ➤ New for the group you want to create.

3  In the New <type> Group dialog box enter a name and description.

4  In the lower pane of the dialog box, check the items you want to add to the group.

**NOTE** If you are using the extended properties feature, and have created User-defined properties, they are displayed. Click the cell in the Value column to assign an attribute.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ right-click <group-name> ➤ New

# Importing Point Files

Use the Import Point File command to import points into the Survey database and network as control points or non-control points.

In the import settings, you have an option for linework processing. The selected point file format determines the column order, (P,N,E,Z,D), the delimiter, and coordinate zone of the file being imported. If you select the Process Linework option, the point file format must contain a Raw Description column. A point file format with a coordinate system zone that is different from the coordinate zone assigned to the Survey Database will require a transformation.

Each time you import a point file, a point group is created on the Prospector tab in Toolspace. The file name for the file you import is the default point group name.

**TIP** You can also use the Import Survey Data wizard to import point files. For more information, see Import Survey Data Wizard (page 239).

**To import a point file**

1  In Toolspace, on the Survey tab, right-click a <named> network.

2  Click Import ➤ Import Point File.

3  Browse to the file you want to import and click Open.

4  Make any necessary changes to the Import Point File settings and click OK.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data 

Toolspace

&lt;named&gt; network ➤ Import ➤ Import Point File

## Importing Points From a Drawing

Use this command to import points from a drawing into the Survey database and network as control points or non-control points.

---

**TIP** You can also use the Import Survey Data wizard to import points from a drawing. For more information, see Import Survey Data Wizard (page 239).

---

**To import points from a drawing**

1 In Toolspace, on the Survey tab, right-click a &lt;named&gt; network.

2 Click Import ➤ Import Points From Drawing.

---

**NOTE** COGO points are imported from the current drawing.

---

3 At the command line, enter one of the following:

■ All: imports all points in the drawing.

■ Numbers: enter the point numbers you want to import.

■ Group: select the point group you want to import.

■ Selection: select points in the drawing.

4 Make any necessary changes in the Import Points From Drawing settings and click OK.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Import Survey Data 

Toolspace

&lt;named&gt; network ➤ Import ➤ Import Points From Drawing

## Import Events

The import event provides a framework that you can use to view and edit specific survey data that is referenced within the import event.

An import event is created each time you import data using either the Import Survey Data wizard, the import commands on the survey network shortcut menu, or the import Survey LandXML command.

Import Events are displayed as a collection within a database on the Toolspace Survey tab. The default name for the import event is the same as the imported file name "&lt;File name&gt;.&lt;ext&gt;".

The Import Event collection contains the Networks, Figures, and Survey Points that are referenced from the specific import command, and provides a convenient way to remove, re-import, and reprocess linework, and insert survey data into the current drawing.

---

**TIP** Commands entered in the Survey Command Window or from using the Run Batch File command do not create an import event. To maintain the integrity of existing networks, you should create a separate network when you use the Survey Command Window.

---

**To edit an import event**

1 In Toolspace, on the Survey tab, expand a named survey database, and then expand the Import Events collection.

2 In the Import Events Editor vista, select an import event, right-click ➤ Edit.

**To delete an import event**

1 In Toolspace, on the Survey tab, expand a named survey database, and then expand the Import Events collection.

2 Select an import event, right-click ➤ Delete.

**To change the order of import events**

1 In Toolspace, on the Survey tab, expand a named survey database, and then expand the Import Events collection.

2 Select the import event and drag it to a new location in the collection.

## Quick Reference

Toolspace

  Survey tab: expand Survey Databases ➤ <named> survey database ➤ Import Events ➤ <named> event

# Import Event Commands

Use the following right-click commands to view and edit data in the Import Event collection and in individual import events.

**Import Event Collection**

■ Import Survey Data: Displays the Import Survey Data wizard.

■ Edit: Displays the Import Events Editor in the panorama vista.

■ Process Linework: Displays the Process Linework dialog box where you can edit settings if necessary and then process the linework for all import events using the specified settings.

■ Insert Into Drawing: Inserts data from all import events into the current drawing.

■ Remove From Drawing: Removes data from the import events created in the current drawing.

**Import Event**

■ Properties: Displays the properties for a selected event, such as name, description, and linework code set.

■ Edit: Displays the Import Events Editor in the panorama vista.

- Re-import: Displays the import settings for the event. Edit settings if needed and click OK to re-import the event data.

- Process Linework: Displays the Process Linework dialog box where you can edit settings if necessary and then process the linework for the selected import events using the specified settings.

# Editing Survey Point Properties

After importing survey data, you can edit survey point descriptions to correct field coding errors.

A survey point is a COGO point that is created by the Survey feature. When the survey point description is changed and the field coding errors are corrected you can re-process the linework. For more information, see Processing Survey Linework (page 251).

**To edit survey point properties**

1 Do one of the following:

- In the drawing, right-click a survey point ➤ Edit Survey Point Properties.

- From any survey point list view right-click a point ➤ Properties.

2 In the Survey Point Properties dialog box, edit the point name and the point description.

3 Click OK. At the command line, you are prompted to select another point. Continue to edit the points as necessary.

4 Press Enter to end the edit command and update the linework.

To process linework at anytime, right-click the Survey Points collection on the Survey tab in Toolspace ➤ Process Linework.

## Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ Survey Point Properties 

Toolspace

Expand Survey Point Collection ➤ right-click a survey point in a point list ➤ Properties

Command Line

EditSvPointProperties

# Processing Survey Linework

Process Linework during the import of survey data or as a post import process.

**IMPORTANT** Figures that are created using a Linework Code Set do not automatically update when you modify survey data by updating figures, changing survey point descriptions, editing point coordinates through changes in observations, or network and traverse adjustment. In order to obtain the most accurate results, it is recommended that you process the linework each time you make modifications to the survey data.

**To process or re-process linework**

1  In Toolspace, on the Survey tab, do one of the following:

   ■ Right-click the Import Events collection ➤ Process Linework. Linework for all import events in the collection is processed in the order that the import events are displayed in the Import Events collection. Use this command in situations where a figure spans across multiple import events. For example, in the first import event EP1 was created, but in a subsequent import event an attempt is made to continue the same EP1.

   ■ Right-click a named import event ➤ Process Linework. Linework for the event you selected is processed.

   ■ Right-click the Survey Points collection ➤ Process Linework. Linework for all points, including points from previous versions and manually entered points.

2  In the Process Linework dialog box, verify the settings and click OK.

## Quick Reference

Toolspace

Right-click Import Event Collection or <named> Import Event ➤ Process Linework

# Adding and Editing Survey Data

<div style="text-align: right; font-size: 3em; font-weight: bold;">11</div>

You can use AutoCAD Civil to add and edit survey data.

To add survey data to your drawing, you can:

- Use the AutoCAD Civil Toolspace Survey tab to define and manage survey data such as point, setups, directions, traverses, and figures.

- Use the AutoCAD Civil Survey Command Window to define survey data such as baselines, centerlines, and intersections, and to view and manage survey command input and output history.

In AutoCAD Civil, a survey database holds all the observed angles and distances for each point.

## Control Points

You can use several methods to create control points.

Control points are points you create at a known location. Control points are not affected by any adjustments or corrections you make. When you create a control point, it is added to survey database.

The Control Points collection on the Toolspace Survey tab expands to display individual control points.

## Creating Control Points

Use the Toolspace Survey tab to create a control point using known northing and easting, or latitude and longitude (if the survey database references a coordinate zone), and elevation coordinates, to be used for traverse and least squares adjustments.

**To create a control point**

1 In Toolspace, on the Survey tab, right-click the Control Points collection. Click New to open the New Control Point dialog box.

2 Enter the point number.

**IMPORTANT** If there is a Survey Point (Non-Control point) with a matching number then the data for that point displays in the New Control Point dialog box. If there is no Survey Point defined in the Survey database that has a matching name, the program checks the active drawing to see if there is an AutoCAD Civil point with a matching number. If there is an AutoCAD Civil point with a matching number, then the point information displays in the New Control Point dialog box.

**3** Optionally, enter the point name.

**IMPORTANT** You can define a point by Name, by deleting the number in the Number field and leaving it blank. Enter a Name in the Name field. If there is already a control point with that name an error message is displayed. If there is a Survey point with a matching name then the data for that point displays in the New Control Point dialog box. If there is no Survey Point or Control point defined in the Survey database that has a matching name, the program checks the active drawing to see if there is an AutoCAD Civil point with a matching name. If there is an AutoCAD Civil point with a matching name, then the point information displays in the New Control Point dialog box.

**4** Enter the easting (X-coordinate) of the new control point.

**5** Enter the northing (Y-coordinate) of the new control point.

**6** Optionally, enter the elevation (Z-coordinate) of the new control point.

**7** Enter a description for the control point.

**8** Optionally, enter the latitude of the point. The Northing property value is calculated from the value entered in this field.

**NOTE** This property is not available if the survey database does not have an assigned coordinate system zone.

**9** Optionally, enter the longitude of the point. The Easting property value is calculated from the value entered in this field.

**NOTE** This property is not available if the survey database does not have an assigned coordinate system zone.

**10** Click OK to create the new control point. The new control point is displayed in the control points list view in Toolspace.

**To create a new control point using the survey command language**

**1** In Toolspace, on the Survey tab, right-click on the network that you want to add the control point to. Click Survey Command Window.

**2** To create a control point by northing and easting, at the Command line, enter:

NE (point) [north] [east] (description)

To create a control point by northing, easting, and elevation, at the Command line, enter:

NEZ (point) [northing] [easting] (elevation) (description)

**3** To create a control point by latitude and longitude (if the survey database settings reference a coordinate zone), enter:

LAT LONG (point) [latitude] [longitude] (description)

**NOTE** For parameter descriptions, see the Quick Reference tab.

**Examples**

The following example sets point 2 at northing 2000, easting 2000 with an elevation of 100, and a description of TP.

*NEZ 2 2000 2000 100 TP*

The following example sets point number 55 at latitude 45°16'17" and longitude 36°55'55".

*LAT LONG 55 45.1617 36.5555*

## Quick Reference

Toolspace

  Survey tab: right-click Control Points ➤ New

Survey Command Window Syntax

  NE (point) [northing] [easting] (description)
  or
  NEZ (point) [northing] [easting] (elevation) (description)
  or
  LAT LONG (point) [latitude] [longitude] (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| northing | The northing of the point. |
| easting | The easting of the point. |
| elevation | The elevation of the point. |
| latitude | The latitude of the point. Enter this value using the current survey database Angle unit. Use a positive value to indicate a location above the equator, or a negative value for one below it. |
| longitude | The longitude of the point. Enter this value using the current survey database Angle unit. Use a positive value to indicate a location east of the prime meridian, or a negative value for a location west of the prime meridian. |
| description | An optional description associated with the point. |

# Editing Control Points

Use the Control Points Editor to view and edit control point data for individual or multiple control points.

For individual control points, the Control Points Editor provides the same functionality as the Control Points properties list view in the Toolspace Survey tab, but it is more suited for horizontal viewing of a tabulation of this type of data. The benefit of editing control points in the Control Points Editor is that you can easily switch to the Survey tab and browse through the data and then return back to the Control Points Editor to continue with your edits.

**To edit control points**

**1** Do one of the following:

- To view and edit all control points, in Toolspace, on the Survey tab, right-click the Control Points collection and click Edit to open the Control Points Editor.

- To view and edit a single control point, in Toolspace on the Survey tab, right-click the control point and click Edit to display the Control Points Editor.

**NOTE** You can also edit a single control point in the Toolspace properties list view, which is displayed when you select a control point in the tree view on the Survey tab.

**2** Modify the applicable fields as required.

**NOTE** When you make a change to any control point data, the entire row is displayed in bold until you save the changes to the survey database.

**3** To edit the values in multiple rows, select the rows, right-click the column heading, click Edit, and enter the new value.

**4** The selected row values for the column are updated with the new value.

**NOTE** If no rows are selected, then all rows are updated. Press the ESC key to clear the selection.

**5** To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload. Alternatively, to undo all the changes in the vista, click 🔄 .

**6** To apply the changes for a single row or for multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click 💾 .

**7** For information about the standard panorama vista functions, see The Panorama Window (page 102).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Control Points ➤ Edit

# Non-Control Points

Use the Survey tab to create non-control points.

Non-control points are created with a northing, easting, and optional elevation.

Non-control points can be promoted to control points if you reference the point as a control point for creating a traverse, or reference the point as a setup to make observations to other points that may affect locations during an analysis.

The Non-Control Points collection on the Toolspace Survey tab expands to display individual non- control points.

# Creating a Non-Control Point

Create a point using known northing, easting, optional latitude and longitude, and optional elevation.

**To create a non-control point**

**1** In Toolspace, on the Survey tab, right-click the Non-Control Points collection and click New to open the New Non-Control Point dialog box.

**2** Enter the point number. You do not need to assign a point number if auto point numbering is on.

> **IMPORTANT** If there is an AutoCAD Civil point with a matching number, then the point information displays in the New Non-Control Point dialog box.

**3** Optionally, enter the point name.

> **IMPORTANT** You can define a point by Name, by deleting the number in the Number field and leaving it blank. Enter a Name in the Name field. If there is already a Control point or Non-control with that name an error message is displayed. If there is a point Survey point with a matching name then the data for that point displays in the New Non-Control Point dialog box. If there is no Survey Point or Non-Control point defined in the Survey database that has a matching name, the program checks the active drawing to see if there is an AutoCAD Civil point with a matching name. If there is an AutoCAD Civil point with a matching name, then the point information displays in the New Non-Control Point dialog box.

**4** Enter the easting (X-coordinate) of the new non-control point.

**5** Enter the northing (Y-coordinate) of the new non-control point.

**6** Optionally, enter the elevation (Z-coordinate) of the new non-control point.

**7** Enter a description for the non-control point.

**8** Enter the latitude of the point. The Northing property value is calculated from the value entered in this field.

> **NOTE** This property is not available if the survey database does not have an assigned coordinate system zone.

**9** Enter the longitude of the point. The Easting property value is calculated from the value entered in this field.

> **NOTE** This property is not available if the survey database does not have an assigned coordinate system zone.

**10** Click OK to create the new non-control point. The new control point is displayed in the non-control points list view in Toolspace.

**To create a non-control point using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network that you want to add control points to. Click Survey Command Window.

**2** At the Command line, enter:

NE SS (point) [north] [east] (elevation) (description)

> **NOTE** For parameter descriptions, see the Quick Reference tab.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Non-Control Points ➤ New

Survey Command Window Syntax

NE SS (point) [North] [East] (elevation) (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| north | The northing of the point. |
| east | The easting of the point. |
| elevation | The elevation of the point. |
| description | An optional description associated with the point. |

## Editing Non-Control Points

Use the Non-Control Points Editor to view and edit point data for individual or multiple non-control points.

For individual points, the Non-Control Points Editor provides the same functionality as the properties list view in Survey tab, but is more suited for horizontal viewing of a tabulation of this type of data. The benefit of editing non-control points in the Non-Control Points Editor is that you can switch to the Survey tab and browse through the data and then return to the Non-Control Points Editor to continue with your edits.

**To edit non-control points**

1  Do one of the following:
  ■  To view and edit all non-control points, on the Survey tab, right-click the Non-control Points collection and click Edit to display the Non-Control Points Editor.

  ■  To view and edit a single non-control point, in the Survey tab, click the Non-Control Points collection. In the item list view, click the point you want to edit.

2  Modify the applicable fields as required.

   **NOTE** When you make a change to any point data, the entire row is displayed in bold until you save the change to the survey database.

3  To edit the values in multiple rows, select the rows, right-click the column heading, click Edit, and enter the new value.

4  The selected row values for the column are updated with the new value.

   **NOTE** If no rows are selected, then all rows are updated.

5  To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload. Alternatively, to undo all the changes in the vista, click  .

**6** To apply the changes for a single row or for multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click ![save icon] .

**7** For information on the standard panorama vista functions, see The Panorama Window (page 102).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network named> ➤ right-click Non-Control Points ➤ Edit

# Setups

Setups are the instrument setups that are defined in the survey database. The Setups collection on the Toolspace Survey tab expands to display individual setups.

# Creating a New Setup

Use the Toolspace Survey tab to create new instrument setups.

You can create a point using an angle turned from either a specified reference line or a previously determined backsight point. Additionally, you can also use a vertical angle or vertical distance to locate the point.

**To create a new setup**

**1** In the Survey Toolspace tab, right-click on the Setups collection and click New to display the New Setup dialog box.

**2** Enter the station point for the setup.

> **NOTE** If you enter a non-existent station point, you are prompted to create a point and the New Control Point dialog box is displayed where you can create a new station point.

**3** Enter the backsight point for the setup. When this column contains a valid entry, the Backsight Direction column displays the calculated backsight direction value and is read-only.

> **NOTE** If you enter a non-existent backsight point, you are prompted to create a point and the New Control Point dialog box is displayed from which you can create a new backsight point.

> **NOTE** Setups can be created using the names of Survey Points that exist in the Survey Database. Enter the point names in the Station Point and Backsight Point fields.

**4** Enter a backsight direction. This field is read-only if the Backsight Point column contains a valid point number.

**5** Enter the backsight orientation angle for the setup.

> **NOTE** This is the reference angle for all angle observations except Face1 and Face2 angles.

**6** Enter the direct angle reference on the backsight for Face1 angles.

**7** Enter the reverse angle reference on the backsight for Face2 angles.

**8** Enter the measured instrument (theodolite) height. This is typically measured from the center of the theodolite optics to the setup point on the ground.

**9** Click OK to create the new setup.

**To create a setup using the survey command language**

**1** In Toolspace, on the Survey tab, right-click on the network that you want to add control points to. Click Survey Command Window.

**2** At the Command line, enter:

STA [point] (instrument height) (description)

> **NOTE** For parameter descriptions, see the Quick Reference tab.

**Example**

This example sets the instrument on point 1. It is 5.1 feet above ground elevation. Therefore, the instrument elevation is 105.1 feet.

NEZ 1 1000 1000 100

AZ 1 2 90

*STA 1 5.1*

!STATION h.i.: 105.10

!POINT: 1 NORTH: 1000.0000 EAST: 1000.0000 ELEV: 100.0000

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Setups ➤ New

Survey Command Window Syntax

STA [point] (instrument height) (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the traverse station. |
| instrument height | The height of the instrument above ground elevation. |
| description | An optional description associated with the point. |

# Defining New Station and Backsight Points

When creating a new setup, you can specify station and backsight points. If you enter a point identifier that does not exist in the survey database, you are prompted to create a new point in the New Control Point dialog box.

**To define new station or backsight points**

**1** In the New Setup dialog box, enter the new station or backsight point in the respective fields.

**2** Click Yes when you are prompted with the message asking you if you want to create the point.

**3** In the New Control Point dialog box, enter the easting, northing, elevation, and description for the new point. Click OK.

**4** The point is added to the Control Points collection and the survey database.

# Viewing and Editing Setups

Use the Toolspace Survey tab to view and edit setup data for multiple setups or for an individual setup.

For individual setups, the Setups Editor vista provides the same functionality as the setup properties list view in the Toolspace Survey tab, but is more suited for horizontal viewing of a tabulation of this type of data. The benefit of editing setups in the Setups Editor is that you can easily switch to the Survey tab and browse through the data and then return to the Setups Editor to continue with your edits.

**To edit setups**

**1** Do one of the following:
- To view and edit all setups, on the Survey tab, right-click the Setups collection and click Edit to display the Setups Editor.

- To view and edit a single setup, on the Survey tab, right-click an individual setup and click Edit to display the Setups Editor.

**NOTE** You can also edit a single setup in the Toolspace item view, which is displayed when you select a setup in the tree view on the Survey tab.

**2** Modify the applicable fields as required.

**NOTE** When you make a change to any setup data, the entire row is displayed in bold until you save the change to the survey database.

**3** To edit the values in multiple rows, select the rows, right-click the column heading, click Edit, and enter the new value.
The selected row values for the column are updated with the new value.

**NOTE** If no rows are selected, then all rows are updated.

**4** To undo the changes for a single row or for multiple rows, select the row(s), right-click, and click Reload. Alternatively, to undo all the changes in the vista, click ⟳ .

**5** To apply the changes for a single row or multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click 💾 .

**6** For information about the standard panorama vista functions, see The Panorama Window (page 102).

**To edit setups that observe a point**

**1** On the Toolspace Survey tab, expand the network name and click Survey Points.

**2** In the Toolspace list view, right-click the point for which you want to edit the setups that observe. Click Edit Setups That Observe to open the Setups Editor.

**3** In the Setups Editor, optionally, right-click a setup and click Edit Observations.

**4**  Modify the applicable fields as required.

> **NOTE** When you make a change to any setup data, the entire row is displayed in bold until you save the change to the survey database.

**5**  To edit the values in multiple rows, select the rows, right-click the column heading, click Edit, and enter the new value.

**6**  The selected row values for the column are updated with the new value.

> **NOTE** If no rows are selected, then all rows are updated.

**7**  To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload. Alternatively, to undo all the changes in the vista, click ⟳ .

**8**  To apply the changes for a single row or for multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click 💾 .

**9**  For information about the standard panorama vista functions, see The Panorama Window (page 102).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Setups ➤ Edit

## Deleting Setups

Delete setups from the Toolspace Survey tab list view or from the Setups Editor vista.

Use the Setups Editor vista to mark setups for deletion, and then accept or roll back the deletions.

> **NOTE** When a setup is deleted, only the setup is deleted, not the points that were created by the setup.

**To delete a setup from the Toolspace Survey tree**

**1**  In Toolspace, on the Survey tab, expand the Setups collection.

**2**  Right-click a setup and click Delete.

**To delete a setup from the Setups Editor**

**1**  In Toolspace, on the Survey tab, right-click the Setups collection and click Edit to open the Setups Editor.

**2**  Select the setup(s) to delete, right-click, and click Delete.

The setup icon is replaced with a ✖ and the row is set to read-only.

**3**  To undo the changes for a single row or for multiple rows, select the row(s), right-click, and click Undelete. Alternatively, to undo all the changes in the vista, click ⟳ .

**4**  To apply the changes for a single row or multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all deletions and changes in the vista, click 💾 .

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Setups ➤ Edit Observations
or
Survey tab: Survey Database ➤ <database name> ➤ Networks ➤ <network name> ➤ Setups ➤ right-click <named setup> ➤ Delete

# Observations

Observations are measured to survey points in relation to a setup.

You use the Toolspace Survey tab and the Observations Editor to add, edit, and manage observations at a specific setup.

In addition to creating and editing observation at a specific setup in the Observations Editor, you can create observations using the Survey Command Window.

Observations are used to measure locations on the Earth's surface that have been determined by surveying (geometric) methods. You can locate points by known coordinates, or in reference to some other known location, such as an angle and distance from an existing point.

When automatic point numbering is off, you must specify the point numbers. If automatic point numbering is on, the points are automatically numbered. Subsequent points are assigned the next available number in sequential order. For information about setting automatic point numbering, see Setting Survey Command Window Options (page 221).

It is helpful to establish a point numbering system for all your projects. A recommended point numbering method might be to number traverse points sequentially, starting with 1. If the job contains fewer than 100 traverse points, then number all sideshots observed from point 1 with numbers 101 to 199, and sideshots taken at point 2 with numbers 201 to 299. This method makes it easy to determine where any given point was observed from.

## Browsing to an Observation

Browse to an observation in the Observation Editor.

In the Observations Editor you have the options to view, create, and edit survey data.

**To browse to an observation**

1  Click Survey tab ➤ Modify panel ➤ Browse To Survey Data Drop-down ➤ Browse To Observation 🛱 .

2  In the Edit Setups That Observe dialog box, enter the point identifier for the observation you wish to find.
The specified observation is displayed in the Observation Editor.

### Quick Reference

Ribbon

Survey tab ➤ Modify panel ➤ Browse To Survey Data Drop-down ➤ Browse To Observation 🛱

Command Line

BrowseToSvObservation

# Enabling Point Numbering

When automatic point numbering is on, new points are automatically assigned the next available point number. When automatic point numbering is off, you must specify new point numbers as they are needed.

**To turn point numbering on or off**

1 In the Toolspace Survey tab, right-click the survey database name and click Edit Survey Database Settings.

2 In the Survey Database Settings dialog box, expand the Survey Command Window property group.

3 Do one of the following:

■ To enable automatic point numbering, select the Auto Point Numbering check box.

■ To turn off automatic point numbering, clear the Auto Point Numbering check box.

4 Click OK.

---

**NOTE** For parameter descriptions, see the Quick Reference tab.

---

**To turn point numbering on or off using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network name for which you want to turn automatic point numbering on or off. Click Survey Command Window.

2 To turn automatic point numbering on, at the Command line, enter:
AUTO ON [point]

3 To turn automatic point numbering off, at the Command line, enter:
AUTO OFF

## Example

The following example shows a valid entry using the Automatic Point Numbering On command with 1001 as the starting point number:

*AP ON 1001*

If you do not enter a point number, then the program assigns the next available point number.

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ right-click <database name> ➤ Edit Survey Database Settings

Survey Command Window Syntax

AUTO ON [point]
or
AUTO OFF

| Parameter | Definition |
| --- | --- |
| point | The assigned starting point number. |

# Creating Observations

Use the Observations Editor to add observations to an existing setup.

Add points using one of the following methods:

■ Angle: Creates an observation using an angle turned from either a specified reference line or a previously determined backsight point. For an example and illustration of this method, see Creating an Observation by Angle and Distance (page 266).

■ Deflection: Creates an observation using a deflection angle and distance from an established line. For an example and illustration of this method, see Creating an Observation by Deflection Angle and Distance (page 270).

■ Face1/Face2 angle: Creates an observation from a traverse station using a horizontal angle measured on Face1 or Face2 of your instrument. For an example and illustration of this method, see Creating an Observation Using a Face1 or Face2 Angle (page 272).

■ Azimuth: Creates an observation from an occupied traverse station by entering a recorded azimuth and distance. For an example and illustration of this method, see Creating an Observation Using an Azimuth (page 275).

■ Bearing: Creates an observation in reference to the occupied point using bearing and distance. For an example and illustration of this method, see Creating an Observation by Bearing and Distance (page 268).

**To add an observation to a setup**

1 In Toolspace, on the Survey tab, right-click the setup name and click Edit Observations.

2 In the Observations Editor, right-click either an existing observation or an empty row and click New.

A new row is added to the Observations Editor.

> **NOTE** If you select an existing row and add a new observation, the new observation uses the settings of the selected row. If you select an empty row, the new observation's settings are based on the Database Settings (page 216).

3 Modify the fields by selecting the field and entering the new value as needed.

4 To edit the Angle Type, Distance Type, Vertical Type, and Target Type fields, click the field and select a new value from the drop-down lists. For information about these types, see Survey Database Settings Dialog Box (page 2378).

> **NOTE** If no rows are selected, then all rows are updated.

5 To create an observation using a point name, delete the value in the point number field and then specify the Angle and Distance values, and then enter a point name. The Angle and Distance values must be entered before you enter the Name. If there is a point Survey point with a matching name then the data for that point is displayed.

6 To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload. Alternatively, to undo all the changes in the vista, click ⟳ .

> **NOTE** If you have new observations and have not applied them, clicking the Reload button will discard the new observations.

7 To apply the changes for a single row or multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click 💾 .

**8** For information about the standard panorama vista functions, see The Panorama Window (page 102).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Setups ➤ Edit Observations

# Creating Observations Using the Survey Command Language

In addition to adding observations using the Observations Editor, you can add observations using the survey command language.

## Creating an Observation by Angle and Distance

Use the angle distance (AD) command to quickly create a point by angle and distance.

Create a point using an angle turned from either a specified reference line or a previously determined backsight point. You can also use a vertical angle or vertical distance to locate the point.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create an observation by angle and distance, using the command language**

**1** In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

**2** At the Command line, enter:

AD (point) [angle] [distance] (description)

**To create an observation by angle, distance, and vertical angle, using the command language**

**1** In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

**2** At the Command line, enter:

AD VA (point) [angle] [distance] [vertical angle] (description)

**To create an observation by angle, distance, and vertical distance, using the command language**

**1** In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

**2** At the Command line, enter:

AD VD (point) [angle] [distance] [vertical distance] (description)

**Examples**

This example creates point 103, which is located by turning a horizontal angle of 80°20'40" right at a distance of 100 units from the instrument point with the description WELL. Because no method of vertical difference determination was given, the elevation is <Null>.

AD 103 80.2040 100 WELL

This example creates point 101, which is located by turning a horizontal angle of 100°20'45" right from the backsight and using a vertical angle of 91°15'15" at a distance of 100 units from the instrument point with the description IP.

AD VA 101 100.2045 100 91.1515 IP

This example creates point 102, which is located by turning a horizontal angle of 90°30'30" right from the backsight and using a vertical distance of -1.5 units at a distance of 200 units from the instrument point with the description EL.

AD VD 102 90.3030 200 -1.5 EL

**Points created with the angle distance command:**
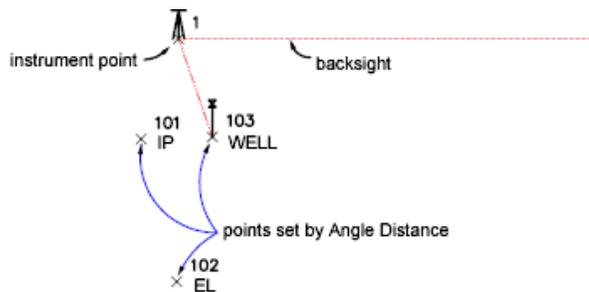


Point 101 is located by turning a horizontal angle of 100°20'45" right from the backsight and using a vertical angle of 91°15'15" at a distance of 100 units from the instrument point with the description IP.

Point 102 is located by turning a horizontal angle of 90°30'30" right from the backsight and using a vertical distance of -1.5 units at a distance of 200 units from the instrument point with the description EL.

Point 103 is located by turning a horizontal angle of 80°20'40" right at a distance of 100 units from the instrument point with the description WELL. Because no method of vertical difference determination was given, the elevation is <Null>.

## Quick Reference

Survey Command Window Syntax

AD (point) [angle] [distance] (description)
or
AD VA (point) [angle] [distance] [vertical angle] (description)
or
AD VD (point) [angle] [distance] [vertical distance] (description)

| Parameter | Definition |
|-----------|-----------|
| point | The assigned point identifier. You do not need to assign a point number if auto point numbering is on. |
| angle | The measured horizontal angle. It is assumed to be clockwise (right). Use a negative number (-) to turn counter-clockwise (left). Express in the current units (Degrees Minutes Seconds (D.MS), grads, decimal degrees (D.DD), radians, or mils). |
| distance | The distance from the instrument point to the point being located. It is measured in the current units unless otherwise specified. Distance is assumed to be a horizontal distance unless VA is used. When VA is used, the distance is recognized as slope distance. |

| Parameter | Definition |
|-----------|------------|
| vertical angle | The direction of the vertical angle (zenith, horizontal, or nadir). Specify the vertical angle in the current angular units. |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |
| description | An optional description associated with the point. |

## Creating an Observation by Bearing and Distance

Use the bearing and distance command (BD) to create an observation in reference to the occupied point using bearing and distance.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create an observation using bearing and distance, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

   BD (point) [bearing] [quadrant] [distance] (description)

**To create an observation using bearing, distance, and vertical angle, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

   BD VA (point) [bearing] [quadrant] [distance] [vertical angle] (description)

**To create an observation using bearing, distance, vertical distance, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

   BD VD (point) [bearing] [quadrant] [distance] [vertical distance] (description)

**Examples**

The following example illustrates using the BD command when the units are set to feet:

NE 1 100 100

STN 1

*BD 2 45 1 100 PT*

! BEARING: N 45-00-00 E DISTANCE: 100.0000

! POINT 2 NORTH: 170.7107 EAST: 170.7107

! ELEV: <Null>

This creates point 2 at the bearing and distance specified.

**Point created with the BD command:**
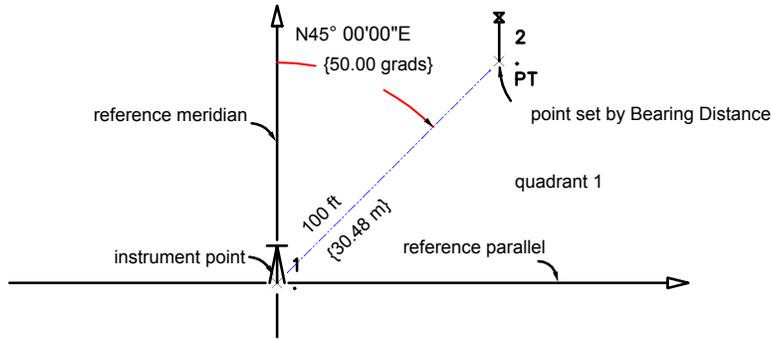


Point 2 is located by turning a bearing of 45°0'0" (50 grads) in Quadrant 1 (NE) at a distance of 100 feet from the instrument point with the description PT.

## Quick Reference

Survey Command Window Syntax

BD (point) [bearing] [quadrant] [distance] (description)
or
BD VA (point) [bearing] [quadrant] [distance] [vertical angle] (description)
or
BD VD (point) [bearing] [quadrant] [distance] [vertical distance] (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| bearing | The bearing. Enter this value in the current angular units (DMS, grads, decimal degrees, mils, or radians). |
| quadrant | The quadrant for the bearing. Specify the quadrant using one of the following numbers: 1 (for NE), 2 (for SE), 3 (for SW), or 4 (for NW). |
| distance | The distance from the instrument point to the point being located. It is measured in the current distance units. The distance is assumed to be a horizontal distance unless VA is used. When VA is used, the distance is recognized as a slope distance. |
| vertical angle | The direction of the vertical angle (zenith, horizontal, or nadir). Type this value in the current angular units (DMS, grads, decimal degrees, mils, or radians). |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |
| description | An optional description associated with the point. |

# Creating an Observation by Deflection Angle and Distance

Use the deflection distance (DD) command to quickly create a point using a deflection angle and distance from an established line.

The term deflection angle refers to the angular direction turned from the extension of the previous course. Additionally, you can also use a vertical angle or vertical distance to locate the point.

---

**NOTE** For parameter descriptions, see the Quick Reference tab.

---

**To create an observation using deflection angle and distance, using the command language**

1. In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2. At the Command line, enter:

    DD (point) [angle] [distance] (description)

**To create an observation using deflection angle, distance, and vertical angle, using the command language**

1. In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2. At the Command line, enter:

    DD VA (point) [angle] [distance] [vertical angle] (description)

**To create an observation using deflection angle, distance, and vertical distance, using the command language**

1. In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2. At the Command line, enter:

    DD VD (point) [angle] [distance] [vertical distance] (description)

**Examples**

The following example illustrates using the deflection distance command.

NE 1 100 100

STN 1

AZ 1 2 100

BS 2

*DD 3 85.2657 60.960*

! AZIMUTH: 385.2657 DISTANCE: 60.9600

! POINT 3 NORTH: 159.3345 EAST: 86.0167

! ELEV: <Null>

**Points created using the DD command:**
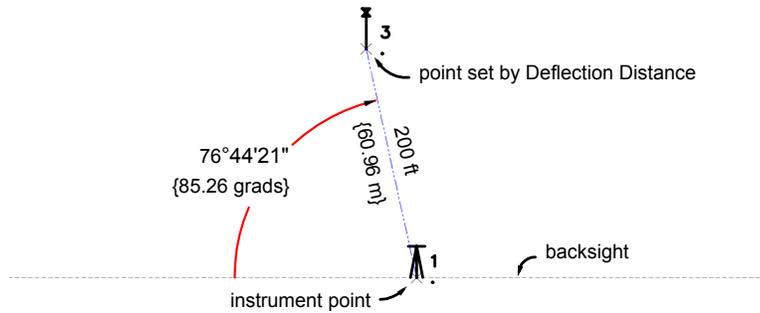


Point 3 is located by turning a deflected angle of 85.2657 right from the extension of the backsight line at a distance of 200. Angles and distances are in the current units, unless you type a qualifying suffix.

## Quick Reference

Survey Command Window Syntax

DD (point) [angle] [distance] (description)
or
DD VA (point) [angle] [distance] [vertical angle] (description)
or
DD VD (point) [angle] [distance] [vertical distance] (description)

| Parameter | Definition |
|-----------|------------|
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| angle | The deflection angle as measured to the point. Type this value in the current angular units (DMS, grads, decimal degrees, radians, or mils). |
| distance | The distance from the instrument point to the point being located. It is measured in the current units unless otherwise specified. The distance is assumed to be a horizontal distance unless VA is used. When VA is used, the distance is recognized as a slope distance. |
| vertical angle | The direction of the vertical angle (zenith, horizontal, or nadir). Type this value in the current angular units (DMS, grads, decimal degrees, mils, or radians). |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |
| description | An optional description associated with the point. |

## Creating an Observation Using a Face1 or Face2 Angle

Create an observation from a traverse station using a horizontal angle measured on Face1 or Face2 of your instrument.

You can then input the measurements, using the Face1 (direct) and Face2 commands. These commands apply collimation (if set), and automatically average the sightings after you input observations.

Face1/Face2 angles can be collected in any order. The only requirement is that the Face1 backsight be recorded first. If Face2 observations are recorded, then a corresponding Face2 backsight should also be recorded. In the case where Face2 foresight observations are recorded previous to the Face2 backsight, the Face2 backsight is assumed to be the Face1 backsight plus 180°.

AutoCAD Civil uses all reciprocal observations to a point (both foresight and backsight) by calculating an average horizontal and vertical distance of all the observations as long as the distance is a non-zero value to calculate the average horizontal and vertical position of a point.

The correct average for all Face1/Face2 and reciprocal observations is applied when the [adjust] command is used. An adjustment may be done to get the correct average values whether there is a closed traverse or not.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create an observation using a Face1 or Face2 angle, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

F1(point) [angle] [distance] (description)

or

F2 (point) [angle] [distance] (description)

**To create an observation using a Face1 or Face2 angle and vertical angle, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

F1 VA (point) [angle] [distance] [vertical angle] (description)

or

F2 VA (point) [angle] [distance] [vertical angle] (description)

**To create an observation using a Face1 or Face2 angle and vertical distance, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:

F1VD (point) [angle] [distance] [vertical distance] (description)

or

F2 VD (point) [angle] [distance] [vertical distance] (description)

**Example: Face1**

NE 2 200 200

AZ 2 1 100

STN 2 1.585

BS 1

*F1 VA 101 0 30.480 100 BS*

! AZIMUTH: 101.6126 DISTANCE: 30.4800

! POINT 101 NORTH: 199.2280 EAST: 230.4702

! ELEV: <Null>

*F1 VA 102 89.5370 15.304 101.2052 IP*

! AZIMUTH: 191.1682 DISTANCE: 15.3013

! POINT 102 NORTH: 184.8457 EAST: 202.1159

! ELEV: <Null>

**Points created Face1 command:**
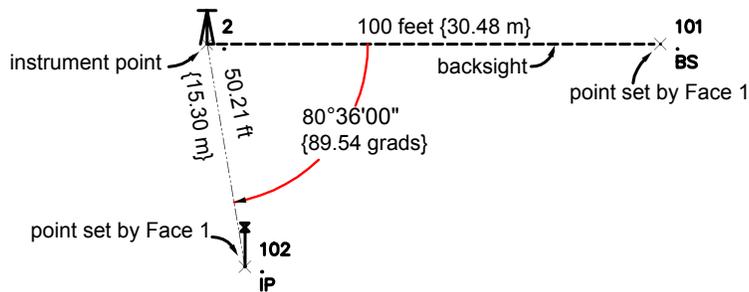


Point 101 is located by turning a horizontal angle of 0 right from Face1, and a vertical angle of 100 at a distance of 30.480 from the instrument point with the description BS.

Point 102 is located by turning a horizontal angle of 89.5370 right from Face1, and a vertical angle of 101.2052 at a distance of 15.304 from the instrument point with the description IP. Angles and distances are in the current units, unless you type a qualifying suffix.

**Example: Face2**

NE 2 200 200

AZ 2 1 100

STN 2 1.585

*F2 1 200*

*F2 VA 101 311.4954 15.30 301.0355 IP*

! AZIMUTH: 213.1080 DISTANCE: 15.300

! POINT 101 NORTH: 198.2901 EAST: 199.6429

! ELEV: <Null>

**Points created Face2 command:**
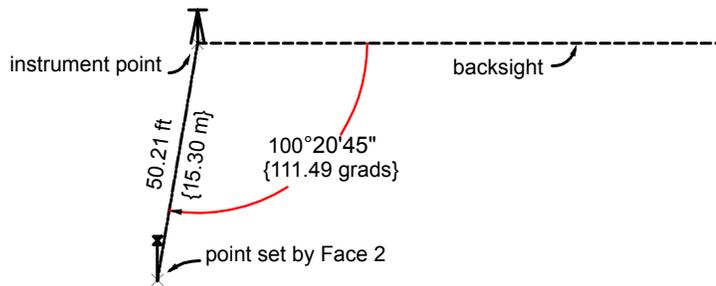


Point 101 is located a distance of 15.30 from the instrument setup at an observed angle right from the backsight of 311.4954. The actual physical angle turned is 111.49, with a vertical angle of 301.03. The description for the point is IP. Angles and distances are in the current units, unless you type a qualifying suffix.

**NOTE** This is an angle turned with the scope inverted. Normally, the backsight is set at a 180° angle, so that the difference between it and the angle is the amount of the angle turned right (100°20'45").

## Quick Reference

Survey Command Window Syntax

F1 (point) [angle] [distance] (description)
F2 (point) [angle] [distance] (description)
or
F1 VA (point) [angle] [distance] [vertical angle] (description]
F2 VA (point) [angle] [distance] [vertical angle] (description]
or
F1 VD (point) [angle] [distance] [vertical distance] (description)
F2 VD (point) [angle] [distance] [vertical distance] (description)

| Parameter | Definition |
|---|---|
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| angle | The measured horizontal angle. It is assumed to be clockwise (right). Use a negative number (-) to turn counter-clockwise (left). Type the horizontal angle in the current angular units (DMS, grads, decimal degrees, radians, or mils). |
| distance | The distance from the instrument point to the point being located. It is measured in the current units unless otherwise specified. The distance is assumed to be a horizontal distance unless VA is used. When VA is used, the distance is recognized as a slope distance. |
| vertical angle | The direction of the vertical angle (zenith, horizontal, or nadir). Type this value in the current angular units (DMS, grads, decimal degrees, mils, or radians). |

| Parameter | Definition |
|---|---|
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |
| description | An optional description associated with the point. If you use a description key, then specific information is assigned to the point. |

## Creating an Observation Using an Azimuth

Create an observation from an occupied traverse station by entering a recorded azimuth and distance.

Additionally, you can also use a vertical angle or vertical distance to locate the point.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create an observation using an azimuth and distance, using the command language**

1   In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2   At the Command line, enter:

ZD (point) [azimuth] [distance] (description)

**To create an observation using an azimuth, distance, and vertical angle, using the command language**

1   In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2   At the Command line, enter:

ZD VA (point) [azimuth] [distance] [vertical angle] (description)

**To create an observation using an azimuth, distance, and vertical distance, using the command language**

1   In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2   At the Command line, enter:

ZD VD (point) [azimuth] [distance] [vertical distance] (description)

## Quick Reference

Survey Command Window Syntax

ZD (point) [azimuth] [distance] (description)
or
ZD VA (point) [azimuth] [distance] [vertical angle] (description)
or

ZD VD (point) [azimuth] [distance] [vertical distance] (description)

| Parameter | Definition |
| --- | --- |
| point | The new point identifier. You do not need to assign a point number if auto point numbering is on. |
| azimuth | The measured azimuth from the traverse station to the point being located. Type this value in the current angular units (DMS, grads, decimal degrees, mils, or radians). |
| distance | The distance from the instrument point to the point being located. It is measured in the current distance units. The distance is assumed to be a horizontal distance unless VA is used. When VA is used, the distance is recognized as a slope distance. |
| vertical angle | The direction of the vertical angle (zenith, horizontal, or nadir). Type this angle in the current angular units. |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |
| description | An optional description associated with the point. |

## Creating a Point By Offset

Create a new point by offsetting the last point you created by a given distance.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a point by offset, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:
   PT OFFSET (point) [offset] [ahead] (description)

**Example**

NEZ 1 1000 1000 30.480

AZ 1 2 100

STN 1 1.585

BS 2

AD 3 100 30.480

*PT OFFSET 4 15.240 7.620*

! AZIMUTH: 224.2238 DISTANCE: 41.0350

! POINT 4 NORTH: 961.9000 EAST: 984.7600

! ELEV: <Null>

Point 4 is located at an offset distance of 15.240, and an ahead distance of 7.620, from point 3.

**Point created by offset:**
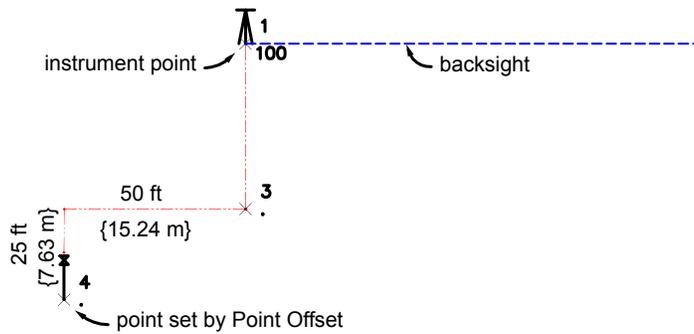


## Quick Reference

Survey Command Window Syntax

PT OFFSET (point) [offset] (ahead) (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| offset | The offset distance from the last point created. |
| ahead | The distance added to the line between the station and the last point created to locate the position of the new point (optional). |
| description | An optional description associated with the point. |

## Creating an Observation Point Using the Stadia Rod

Use the stadia rod intercept method to reduce a distance that has been measured.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create an observation using the stadia rod method, using the command language**

1  In Toolspace, on the Survey tab, right-click the network that you want to add points to, and click Survey Command Window.

2  At the Command line, enter:
    STADIA (point) [angle] [distance] [rod] (vertical angle) (description)

### Quick Reference

Survey Command Window Syntax

STADIA (point) [angle] [distance] [rod] (vertical angle) (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the new point. You do not need to assign a point number if auto point numbering is on. |
| angle | The measured horizontal angle. It is assumed to be clockwise (right). Use a negative number (-) to turn counter-clockwise (left). Type the angle in the current units (DMS, grads, decimal degrees, radians, or mils). |
| distance | The distance from the instrument point to the point being located. It is measured in the current units unless otherwise specified. The distance is assumed to be a horizontal distance unless a vertical angle is specified. When a vertical angle is specified, the distance is recognized as a slope distance. The distance required here is actually the rod intercept multiplied by the stadia constant. For example, if 1.37 is intercepted on the stadia board, and the stadia constant is 100, then you should type 137 as the distance. |
| rod | The rod reading taken by the central hairline of the instrument. This is, in effect, the height above ground of the sighted target point. |
| vertical angle | The direction of the vertical angle to the rod (zenith, horizontal, or nadir) and must be specified in the current angular units. |
| description | An optional description associated with the point. |

## Editing Observations

Use the Observations Editor panorama vista to edit the observations of a selected setup item in the Survey tree view.

**NOTE** When changes made in the Observations Editor are saved back to the survey database, the affected points in the drawing are updated from the survey database.

**To edit observations for a setup**

1 In the Toolspace Survey tab, right-click the named setup and click Edit Observations.

2 Modify the fields by selecting the field and entering the new value.

**NOTE** If an observation is marked with a ⚠ , the observation has been modified outside of the Observations Editor and must be reloaded (by right-clicking and clicking Reload) before you can make any edits to the observation.

**3** To edit the Angle Type, Distance Type, Vertical Type, and Target Type fields, click the field and select a new value from the drop-down lists. For information about these types, see Database Settings (page 216).

**4** To edit the values in multiple rows, select the rows, right-click the column heading, click Edit, and enter the new value.

The selected row values for the column are updated with the new value. If no rows are selected, then all rows are updated.

**NOTE** When you make a change to any observation data, the entire row is displayed in bold until you save the change to the survey database.

**5** To delete observations, click the observation(s) to delete, right-click, and click Delete. The observation icon is replaced with a ✖ and the row is set to read-only.

**6** To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload or Undelete. Alternatively, to undo all the changes in the vista, click ♻ .

**7** To apply the changes for a single row or multiple rows, select the row(s), right-click, and click Apply. Alternatively, to apply all changes in the vista, click 🖫 .

**8** For information about the standard panorama vista functions, see The Panorama Window (page 102).

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Setups ➤ Edit Observations

## Modifying the Description of a Point at the Survey Command Line

You can assign or edit the description of a point.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To modify the description for a point, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network that you to edit, and click Survey Command Window.

**2** At the Command line, enter:

MOD DESC [point 1] [description]

**Example**

NE 1 1000 1000 FENCE

*MOD DESC 1 HOUSE*

! POINT 1 DESC: HOUSE

This example changes the description of point 1 from FENCE to HOUSE.

### Quick Reference

Survey Command Window Syntax

MOD DESC [point 1] [description]

| Parameter | Definition |
|-----------|------------|
| point 1 | The point number which has the description you want to modify. |
| description | An optional description associated with the point. |

## Modifying the Elevation of Points at the Survey Command Line

Use the Survey Command Window to modify the elevation of points.

The elevation modification commands are intended to work only with control and non-control points.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To modify the elevation of a single point**

1 In Toolspace, on the Survey tab, right-click the network that you want to edit, and click Survey Command Window.

2 At the Command line, enter:

MOD ELEV [point 1] (elevation)

**To modify the elevation of a multiple points**

1 In Toolspace, on the Survey tab, right-click the network that you want to edit, and click Survey Command Window.

2 At the Command line, enter:

MOD ELEVS [point 1] [point 2] [elevation]

**To modify point elevation by an increment**

1 In Toolspace, on the Survey tab, right-click the network that you want to edit, and click Survey Command Window.

2 At the Command line, enter:

MOD ELEV BY [point 1] (amount)

or

MOD ELEVS BY [point 1] [point 2] (amount)

**Example**

This example modifies the elevation of point 2 from 33.528 to 32.004 and then increases the elevation of point 2 by an increment of 10.0. Elevations are in the current units, unless you type a qualifying suffix:

NEZ 1 100 100 30.480

NEZ 2 200 200 33.528

*MOD ELEV 2 32.004*

! POINT 2 ELEVATION: 32.004

MOD ELEV BY 2 10.0

! POINT 2 ELEVATION ADDED: 42.004

## Quick Reference

Survey Command Window Syntax

MOD ELEV [point 1] (elevation)
or
MOD ELEVS [point 1] [point 2] [elevation]
or
MOD ELEV BY [point 1] (amount)
or
MOD ELEVS BY [point 1] [point 2] (amount)

| Parameter | Definition |
| --- | --- |
| point 1 | The point number to assign an elevation to. |
| point 2 | Used with point 1 to specify the range of points affected. |
| elevation | The new elevation value. |
| amount | The amount to increment or decrement the height of point 1. To decrement the height, type a negative value. |

# Inserting, Removing, and Updating Points

You can access a standard set point manipulation commands from the Toolspace Survey tab collections and items.

The point manipulation commands are available via a right-click menu and are applicable to the points relative to where the commands are accessed. For example, if you use the Insert into Drawing command on an individual control point, it only inserts that control point.

| Select this... | If you want to do this... |
| --- | --- |
| Update | Update the points in the drawing to the values in the survey database. |
| Insert Into Drawing | Insert points referenced in the survey database into the drawing. |
| Remove From Drawing | Remove points referenced in the survey database from the drawing. |

# Updating the Drawing Points with the Survey Points

If the points in the drawing are out-of-sync with the survey database, you can update them to the values in the survey database.

**To update the drawing points with the survey database points**

**1** In Toolspace, on the Survey tab, right-click the collection or item for which you want to update the points with the survey database.

**2** Click Points ➤ Update.

The points in the drawing and point database are updated with the values from the survey database. Points that are not referenced in the survey database are not affected by the Update command.

The points will not be updated if the points reside on a locked layer.

## Quick Reference

Toolspace

Survey tab: right-click <collection or item> ➤ Points ➤ Update

## Inserting Points into a Drawing

You can insert points into a drawing from the point database. The drawing points, which are referenced by the points in the survey database for the applicable collection or item, are inserted.

**To insert survey points into the drawing**

**1** In the Survey tab, right-click the collection or item for which you want the points (referenced in the survey database) inserted, and click Points ➤ Insert Into Drawing. If you select points that are already in the drawing, then the Duplicate Point Number dialog box is displayed.

**2** Click one of the following options:
   - Click Skip to leave the existing point in the drawing.
   - Click Replace to replace the existing point in the drawing with the selected point in the point database.
   - Click Skip All to leave all existing points in the drawing.
   - Click Replace All to replace the existing points in the drawing with the selected points in the point database.
   - Click Cancel to cancel the command.

## Quick Reference

Toolspace

Survey tab: right-click a collection or item ➤ Points ➤ Insert Into Drawing

## Removing Points from a Drawing

You can remove points that are referenced by the survey database from the drawing.

**To remove points from the drawing**

➤ In the Survey tab, right-click the collection or item for which you want the points (referenced in the survey database) removed from the drawing, and click Points ➤ Remove From Drawing.

## Quick Reference

Toolspace

Survey tab: right-click a collection or item ➤ Points ➤ Remove From Drawing

# Directions

A direction is an angle measured from the north or south meridian.

A direction can be formatted as either a bearing or azimuth and must be defined from an existing control point to another point (either a false point or another control point).

A bearing is less than 90 degrees (PI/2 radians or 100 grads), and is measured clockwise in the NE quadrant, counter-clockwise in the SE quadrant, clockwise in the SW quadrant, and counter-clockwise in the NW quadrant.
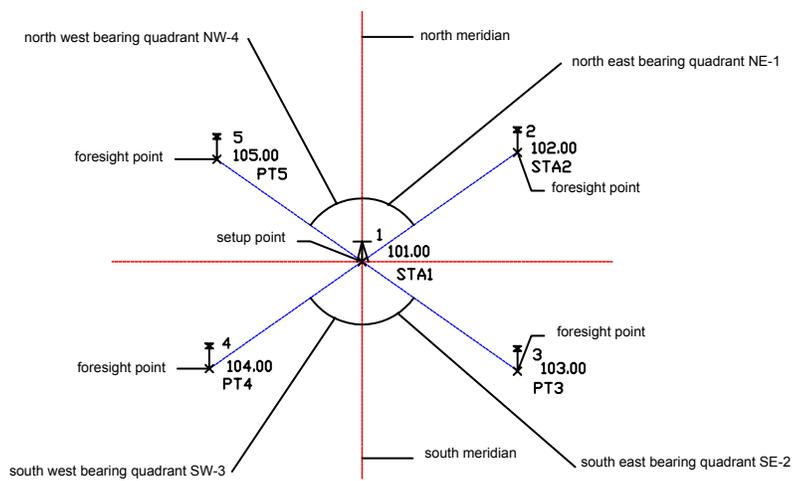


An azimuth is measured clockwise from either the north or south meridian and is always less than or equal to 2PI.
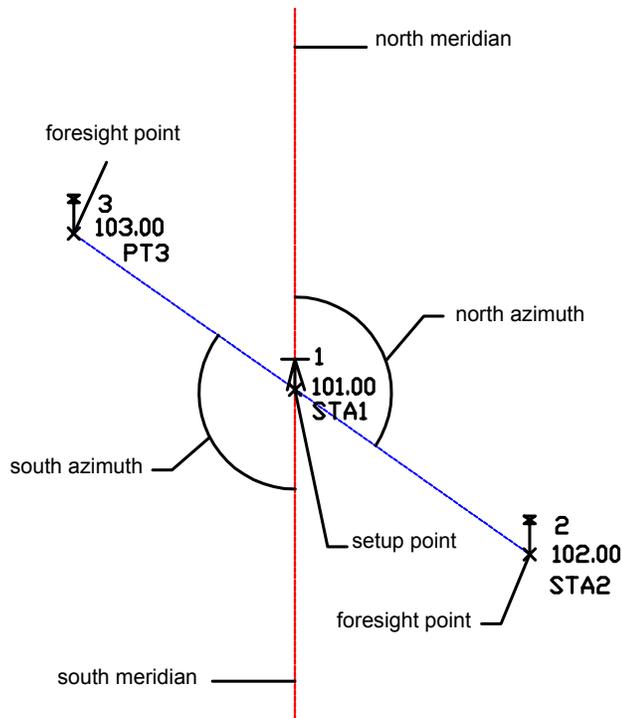
## Creating a Direction

Use the Toolspace Survey tab to create directions.

**To create a direction**

1  In the Toolspace Survey tab, right-click the Directions collection and click New.

2  In the New Direction dialog box, specify the point and direction information. For information about the direction properties, see New Direction/Direction Properties Dialog Box (page 2417).

3  Click OK to create the direction.

### Quick Reference

Toolspace

Survey tab: Expand Survey Databases ➤ <database name> ➤ Networks ➤ <network name> ➤ right-click Directions ➤ New

## Editing a Direction

Use the Toolspace Survey tab item view to edit directions.

You can change the direction value and type.

**To edit a direction**

**1** In the Toolspace Survey tab, click the Directions collection.

**2** In the Survey tab item view, click the field for the direction that you want to edit. Editable fields are colored white.

For information about common functions available in the list view, see Survey Tab Item View (page 2383).

### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ expand <named project> ➤ click Directions ➤ edit in the item view

# Baselines

You can create baselines, which are reference lines from a baseline start point and a baseline end point.

You can create or list points in respect to the defined baseline. The Inverse Point command determines the station and offset distance of an existing point, or group of points, from an existing baseline. The Station/Offset command creates a new point using a station and an offset distance in reference to an existing baseline. Before you can use the Inverse Point, or the Station/Offset commands, you must define the current baseline. If you haven't defined the baseline and you select one of these commands, you are prompted to define the baseline.

# Defining a Baseline

Use the Survey Command Window to define a baseline.

You can create a baseline as a reference line, and then create or list points in relation to the line.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To define a baseline from the Survey Command Window menu**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** Click Baseline menu ➤ Define Baseline.

**3** Enter the point identifier for the origin of the baseline.

**4** Enter the point identifier for the end point of the baseline.

**5** Enter the point identifier for the starting station of the baseline.

**To define a baseline, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** At the Command line, enter:

BL IS [point 1] [point 2] [station]

**Command Line Example**

NEZ 1 1000 1000 30.48

NEZ 2 1500 1500 32.004

*BL IS 1 2 100*

The baseline runs from point 1 to point 2. Point 1 is set as station 1+00. Direction and distances are in the current units, unless you type a qualifying suffix.

**Baseline from point 1 to point 2:**
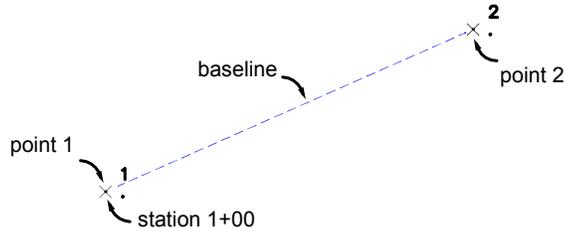


## Quick Reference

Survey Command Window

  Baseline ➤ Define Baseline

Survey Command Window Syntax

  BL IS [point 1] [point 2] [station]

| Parameter | Definition |
|-----------|------------|
| point 1 | The existing point identifier for the beginning point of the baseline. This point is assigned the station number given for the station component. |
| point 2 | The existing point identifier for the end point on the baseline. When you assign this point, you define the direction and length of the baseline. |
| station | The station for the assigned point number. Type the station in the current distance units. Some examples of station entries are 0, 50, and 100. For example, station 150 indicates that the station is 150 feet or meters along the baseline. |

# Listing a Baseline Direction Using Inverse Point

You can determine the station and offset distance of an existing point, or group of points, from an existing baseline with the Inverse Point command.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To inverse a point from the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2  Click Baseline menu ➤ Inverse Point.

**3** Type the start and end of the point range for which you want the station and offset information.

> **NOTE** You must define the baseline as a prerequisite.

**To inverse a point, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** At the Command line, enter:
BL INV [point 1] (point 2)

**Command Line Example**

NE 1 1000 1000

STN 1

ZD 2 90 400

BL IS 1 2 100

BL PT 3 220 50

BL PT 4 270 -50

*BL INV 1 4*

This station and offset of points 1, 2, 3, and 4 are listed:

! Point Station Offset Elevation. Northing Easting

! 1 100.00 0.00 <Null> 1000.00 1000.00

! 2 500.00 0.00 <Null> 1000.00 1400.00

! 3 220.00 50.00 <Null> 950.00 1120.00

! 4 270.00 -50.00 <Null> 1050.00 1170.00

## Quick Reference

Survey Command Window

  Baseline ➤ Inverse Point

Survey Command Window Syntax

  BL INV [point 1] (point 2)

| Parameter | Definition |
| --- | --- |
| point 1 | An existing point in the drawing. It does not need to be adjacent to the baseline since the baseline is infinite in length. |
| point 2 | Used with point 1 to specify a range of points that you want to list. If you do not specify point 2, then only point 1 is listed. |

# Creating a New Point from a Baseline Using a Station and Offset

You can create a new point by specifying a station on a baseline, and an offset distance in reference to an existing baseline, with the Station/Offset command.

**To create a point by station and offset from the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Baseline menu ➤ Station/Offset.

3   Enter the point identifier of the point you want to create.

**NOTE** If Auto point numbering is on, then the next point number is displayed.

4   Enter the station of the new point. The station can be a positive or a negative value.

5   Enter the offset of the new point. The offset can be a positive or a negative value.

6   Optionally, enter a description of the new point.

**To create a point at an offset from a baseline, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    BL PT (point 1) [station] (offset) (description)

**Command Line Example**

NE 1 1000 1000

STN 1

ZD 2 100 121.92

BL IS 1 2 0

*BL PT 3 220 15.24*

! POINT 3 NORTH: 946.7889 EAST: 1214.0113 ELEV:<Null>

*BL PT 4 270 -15.24*

! POINT 4 NORTH: 968.1235 EAST: 1268.5445 ELEV: <Null>

Point 3 is offset a distance of 15.24 to the right of the baseline at Station 2+20 and Point 4 is offset a distance of 15.24 to the left of the baseline at Station 2+70. Distances are in the current units, unless you type a qualifying suffix.

**Baseline created at an offset distance:**
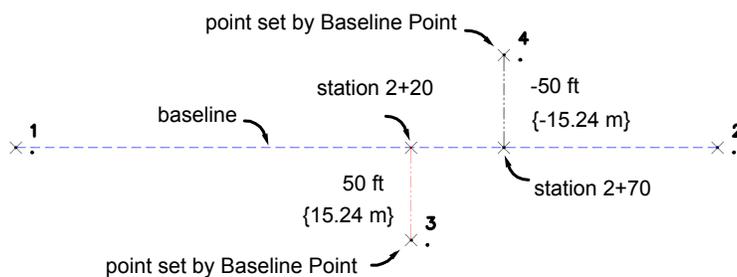
### Quick Reference

Survey Command Window

　Baseline ➤ Station/Offset

Survey Command Window Syntax

　BL PT (point 1) [station] (offset) (description)

| Parameter | Definition |
| --- | --- |
| point 1 | The point identifier for the new point. You do not need to assign a point number if auto point numbering is on. |
| station | The station location of the assigned point. Type the station in the current distance units. Some examples of station entries are 0, 50, and 100. For example, station 150 indicates that the station is 150 feet or meters along the baseline. |
| offset | An optional distance offset from the baseline. |
| description | An optional description associated with the point. |

# Centerlines

A centerline is defined from a figure. After you define the current centerline and current cross section, you can create new points in relation to them.

Before using the Inverse Point command, you must define the current centerline. If you haven't defined the centerline, you are prompted to define the centerline.

The cross section commands operate in a similar manner; if you have not set the current cross section, then you are prompted to set the cross section.

If you have not set the height of instrument and you use either the Station/ Offset Vertical Distance or the Station/Offset Rod command, then the Set Height of Instrument command is automatically invoked. Likewise, the Offset Rod or Offset Vertical Distance commands invoke the Prism command the first time you select them.

# Defining a Centerline

Use the Survey Command Window to quickly define a centerline.

To define a centerline, you specify a figure name, starting station, and starting point number.

If you use a point off the centerline to determine the beginning station, then a perpendicular line is established and that intersection is the starting station. Therefore, any point you use to establish the beginning station must be adjacent to some leg of the centerline.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To define a centerline from the Survey Command Window menu**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Define Figure Centerline.

3 Enter a figure name for the centerline.

**4** Enter the starting station value.

**5** Enter the starting station point identifier. This is the location to start stationing from. The starting station is the perpendicular intersection of the starting station point identifier and the centerline.

**To define a centerline, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** At the Command line, enter:

CL IS [figure] (station) (point)

**Command Line Example**

In the following example, the beginning of this command sequence describes how to draw the centerline figure.

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

NE 4 1100 1100

*CL IS CL1 500 4*

! Figure CL1 is the current centerline.

! Start Station: 400.00 End Station: 1260.56

Figure CL1 is the current centerline. Assigning Point 4 a station of 5+00 results in the centerline starting at station 4+00.

**Centerline figure:**
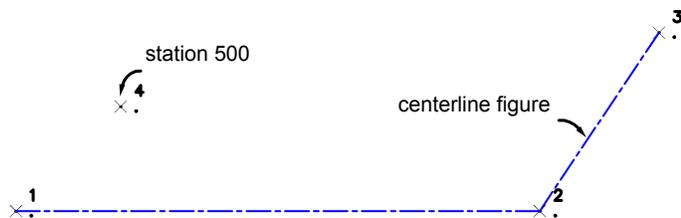


## Quick Reference

Survey Command Window

Centerline ➤ Define Figure Centerline

Survey Command Window Syntax

CL IS [figure] (station) (point)

| Parameter | Definition |
|-----------|------------|
| figure | The figure that you want to use as the centerline. |
| station | The defining station of the centerline. If you do not type a point number, then this is the station at the beginning of the figure. If you type a point number, then the station is at the point you specify. When you don't specify a station, the default 0+00 is used. |
| point | An optional reference point for the station. The point specified need not be on the centerline, but it must be adjacent to the centerline (not beyond the end points). |

# Listing Centerline Data Using Inverse Point

You can display the station, offset, and elevation of an existing point(s) in reference to the current centerline.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To display point data in reference to the current centerline from the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Centerline menu ➤ Inverse Point.

3   Enter the point number range you want the station, offset, and elevation information for.

**To display point data in reference to the current centerline, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    CL INV [point 1] (point 2)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

CL PT 4 200 60.96

CL PT 5 625 -45.72

*CL INV 1 5*

! Point Station Offset Elevation Northing Easting

! 1 0.00 0.00 <Null> 1000.0000 1000.0000

! 2 500.00 0.00 <Null> 1000.0000 1500.0000

! 3 860.56 0.00 <Null> 1300.0000 1700.0000

! 4 200.00 60.96 <Null> 939.0400 1200.0000

! 5 625.00 -45.72 <Null> 1129.3672 1531.2962

These examples return station information along the centerline. Distances are in the current units, unless you type a qualifying suffix.

### Quick Reference

Survey Command Window

  Centerline ➤ Inverse Point

Survey Command Window Syntax

  CL INV [point 1] (point 2)

| Parameter | Definition |
|-----------|------------|
| point 1 | The point to inverse. |
| point 2 | Used with point 1 to specify the range of points affected. This point is optional. If you do not specify point 2, then only data for point 1 displays. |

## Setting the Height of the Instrument

You can establish the instrument height to use with the Station/Offset Rod or Offset Rod commands by using the Set Height of Instrument command.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To set the instrument height from the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Centerline menu ➤ Set Height of Instrument.

3   Enter the instrument height.

   **NOTE** The Set Height of Instrument command does not affect the height of the traverse station. It lets you process level data.

**To set the instrument height, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    HI [elevation]

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

*HI 200*

! Current HI: 200.000

The height of instrument is set to 200.00.

### Quick Reference

Survey Command Window Menu

Centerline ➤ Set Height of Instrument

Survey Command Window Syntax

HI PT [elevation]

| Parameter | Definition |
| --- | --- |
| elevation | The elevation of the optical axis of the instrument above datum (not the height above ground). |

## Creating a New Point by Station Offset

You can create a new point using an offset from a station on the current centerline.

NOTE For parameter descriptions, see the Quick Reference tab.

**To create a point by station offset in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Station/Offset.

3 Enter the point identifier of the point you want to create.

NOTE If Auto point numbering is on, then the next point number is displayed.

4 Enter the station of the new point. The station can be a positive or a negative value.

5 Enter the offset of the new point. The offset can be a positive or a negative value.

6 Optionally, enter a description for the point.

**To create a point at an offset, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:
CL PT (point 1) [station] (offset) (skew angle) (description)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

*CL PT 100 200*

! POINT 100 NORTH: 1000.0000 EAST: 1200.0000 ELEV: <Null>

*CL PT 101 450 -91.44*

! POINT 101 NORTH: 1091.4400 EAST: 1450.0000 ELEV: <Null>

102 600 152.4 50

! POINT 102 NORTH: 931.7283 EAST: 1572.2198 ELEV: <Null>

Point 100 is located at station 200 with a zero offset. Point 101 is located at station 450 with a left offset distance of 91.44. Point 102 is located at station 600 with a right offset distance of 152.4 and a skew angle of 50.

---

**NOTE** The point 102 input uses the Ditto command. Angles and distances are in the current units, unless you type a qualifying suffix.

---

**Centerline point created by a station offset:**
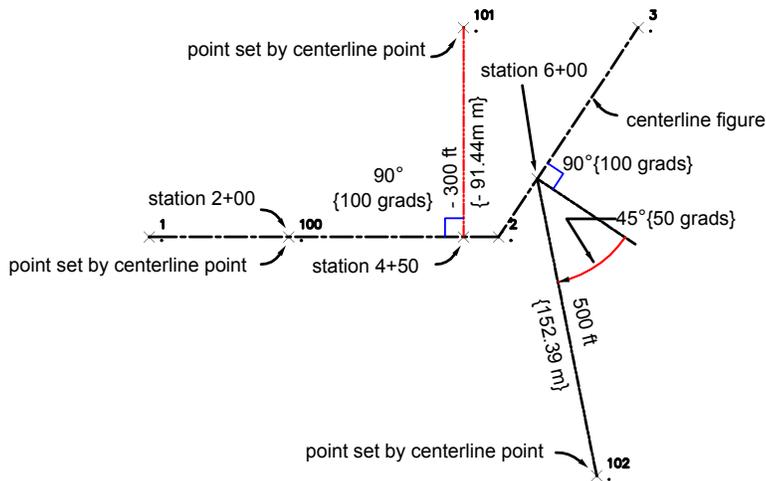


## Quick Reference

Survey Command Window Menu

Centerline ➤ Station/Offset

Survey Command Window Syntax

CL PT (point 1) [station] (offset) (skew angle) (description)

| Parameter | Definition |
|---|---|
| point 1 | The point identifier for the new point. You do not need to assign a point number if auto point numbering is on. |
| station | The station at which to set the point. |
| offset | An optional component that specifies the distance from the centerline to the new point. A negative offset sets the point to left of the centerline, and a positive offset sets it to the right. |
| skew angle | An optional component that specifies the clockwise angle to skew the point from the centerline to create non-perpendicular offsets. Type this value in the current angular units (DMS, grads, decimal degrees, radians, or mils). |
| description | An optional description associated with the point. |

# Creating a Point Using a Station, Offset, and Elevation

You can create a point at a given station, offset, and elevation with respect to the current centerline.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a point by station, offset, and elevation from the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Station/Offset Elevation.

3 Enter the point number of the point you want to create.

**NOTE** If Auto point numbering is on, then the next point number is displayed.

4 Enter the station of the new point. The station can be a positive or a negative value.

5 Enter the offset of the new point. The offset can be a positive or a negative value.

6 Enter the elevation for the point.

7 Optionally, enter a description for the point.

**To create a point by station, offset, and elevation, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:
CL EL (point) [station] [offset] [elevation] (description)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

*CL EL 100 350 45.72 58.7502*

! POINT 100 NORTH: 954.2800 EAST: 1350.0000 ELEV: 58.7502

## Quick Reference

Survey Command Window Menu

Centerline ➤ Station/Offset Elevation

Survey Command Window Syntax

CL EL (point) [station] [offset] [elevation] (description)

| Parameter | Definition |
| --- | --- |
| point | The identifier of the new point that you want to create. You do not need to assign a point number if auto point numbering is on |
| station | The station of the new point. |
| offset | The perpendicular distance from the centerline to the new point. |
| elevation | The elevation of the new point. Elevations are in the current units, unless you type a qualifying suffix. |
| description | An optional description associated with the point. |

## Creating a New Point by Station, Offset, and Rod Height

You can create a point by specifying a station, offset, and rod height from the current centerline. The elevation is computed by algebraically adding the rod reading to the current value set with the Set Height of Instrument command.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a point along a centerline using a rod reading from the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Station/Offset Rod.

3 Enter the station.

4 Enter the offset.

5 Enter the rod reading.

6 Optionally, enter a description for the point.

**To create a point along a centerline using a rod reading, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:
CL ROD (point) [station] [offset] [rod] (description)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

HI 60.96

*CL ROD 100 350 45.72 4.2098*

! POINT 100 NORTH: 954.2800 EAST: 1350.0000 ELEV: 56.7502

## Quick Reference

Survey Command Window Menu

Centerline ➤ Station/Offset Rod

Survey Command Window Syntax

CL ROD (point) [station] [offset] [rod] (description)

| Parameter | Definition |
| --- | --- |
| point | The point identifier for the new point. You do not need to assign a point number if auto point numbering is on. |
| station | The station of the new point. |
| offset | The perpendicular distance from the centerline to the new point. |
| rod | The rod height. This value is subtracted from the height of instrument elevation to determine the elevation of the new point. |
| description | An optional description associated with the point. |

## Creating a New Point by Station, Offset, and Vertical Distance

You can create a point given a vertical distance, with respect to the current centerline, height of instrument, and prism height.

The elevation is computed by subtracting the prism height from the instrument height, then algebraically adding the vertical distance.

**To create a point along a centerline using a vertical distance from the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Centerline menu ➤ Station/Offset Vertical Distance.

3   Enter the point number of the point you want to create.

   **NOTE** If auto point numbering is on, then the next point is displayed.

4   Enter the station of the new point. The station can be a positive or a negative value.

5   Enter the offset of the new point. The offset can be a positive or a negative value.

6   Enter the vertical distance.

7   Optionally, enter a description for the point.

**To create a point along a centerline using a vertical distance, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
   CL VD (point) [station] [offset] [vertical distance]

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

HI 200

PRISM 5

*CL VD 10 350 150 7.25*

! POINT 10 NORTH: 850.0000 EAST: 1350.0000 ELEV: 202.2500

## Quick Reference

Survey Command Window Menu

   Centerline ➤ Station/Offset Vertical Distance

Survey Command Window Syntax

CL VD (point) [station] [offset] [vertical distance]

| Parameter | Definition |
| --- | --- |
| point | The identifier of the new point that you want to create. You do not need to assign a point number if auto point numbering is on. |
| station | The station of the new point. |
| offset | The perpendicular distance from the centerline to the new point. |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |

# Setting the Current Cross Section

You can set the current cross section by station value. When the current cross section is set, you can create new points from this occupied point by supplying the offset and elevation information.

To use the Offset Rod, Offset Elevation, and Offset Vertical Distance commands, you must set the current cross section station.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To set the current cross section from the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Centerline menu ➤ Set Cross Section.

3   Type the station of the current cross section.

**To set the current cross section, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    XS [station]

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

*XS 350*

Current Cross-section Station: 350.00

### Quick Reference

Survey Command Window Menu

Centerline ➤ Set Cross Section

Survey Command Window Syntax

XS [station]

| Parameter | Definition |
|-----------|------------|
| station | The station to use for subsequent cross section commands. |

## Creating a New Point on the Cross Section by Offset and Elevation

You can create a new point by giving an offset and elevation, with respect to the current cross section located on the current centerline.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a point with offset and elevation from the current cross section from the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Offset Elevation.

3 Type the point number of the point you want to set.

4 Enter the offset.

5 Enter the elevation.

**To create a point with offset and elevation from the current cross section using the survey command language**

1 In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

2 At the Command line, enter:
XS EL (point) [offset] [elevation] (description)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

XS 350

*XS EL 100 45.72 58.7502*

! POINT 100 NORTH: 954.2800 EAST: 1350.0000 ELEV: 58.7502

Point 100 is offset from cross section 350 by a distance of 45.72 with an elevation of 58.7502. Distance and elevations are in the current units unless you type a qualifying suffix.

### Quick Reference

Survey Command Window Menu

  Centerline ➤ Offset Elevation

Survey Command Window Syntax

  XS EL (point) [offset] [elevation] (description)

| Parameter | Definition |
|---|---|
| point | The identifier of the new point that you want to create. You do not need to assign a point number if auto point numbering is on. |
| offset | The perpendicular distance from the centerline to the new point. |
| elevation | The elevation of the new point |
| description | An optional description associated with the point. |

## Creating a New Point on the Cross Section by Offset and Rod Height

You can create a point by giving an offset and rod height with respect to the current cross section, height of instrument, and centerline.

The elevation is computed by subtracting the rod from the current value you set with the Set Height of Instrument command.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a point by offset and rod height from the current cross section in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Offset Rod.

3 Enter the point number of the point you want to set.

4 Enter the offset of the new point. The offset can be a positive or a negative value.

5 Enter the rod height.

**To create a point by offset and rod height from the current cross section, using the survey command language**

1 In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

2 At the Command line, enter:

  XS ROD (point) [offset] [rod] (description)

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

XS 350

HI 200

*XS ROD 100 150 7.25*

! POINT 100 NORTH: 850.000 EAST: 1350.000 ELEV: 192.750

Point 100 is offset from cross section 350 a distance of 150 with an elevation of 192.75 (height of instrument - rod).

## Quick Reference

Survey Command Window Menu

Centerline ➤ Offset Rod

Survey Command Window Syntax

XS ROD (point) [offset] [rod] (description)

| Parameter | Definition |
|---|---|
| point | The identifier of the new point that you want to create. You do not need to assign a point number if auto point numbering is on. |
| offset | The distance from the cross section to the new point. |
| rod | The rod height. This value is subtracted from the height of instrument elevation to determine the elevation of the new point. |
| description | An optional description associated with the point. If you use a description key, then specific information is assigned to the point. |

## Creating a New Point on the Cross Section by Vertical Distance

You can create a point by specifying a vertical distance from the current cross section, height of instrument, prism height, and centerline.

The elevation is computed by subtracting the prism height from the value you specified with the Set Height Of Instrument command, and then adding the vertical distance.

NOTE For parameter descriptions, see the Quick Reference tab.

**To create a point by offset and vertical distance from the current cross section in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

2 Click Centerline menu ➤ Offset Vertical Distance.

3 Enter the point number of the point you want to create.

**4** Enter the offset of the new point. The offset can be a positive or a negative value.

**5** Enter the vertical distance.

**To create a point by offset and vertical distance from the current cross section, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click on the network to edit, and click Survey Command Window.

**2** At the Command line, enter:

XS VD (point) (offset) [vertical distance]

**Command Line Example**

BEGIN CL1

NE 1 1000 1000

NE 2 1000 1500

NE 3 1300 1700

END

CL IS CL1

XS 350

HI 200

PRISM 5

*XS VD 100 150 7.25*

! POINT 100 NORTH: 850.000 EAST: 1350.000

! ELEV: 202.2500

## Quick Reference

Survey Command Window Menu

Centerline ➤ Offset Vertical Distance

Survey Command Window Syntax

XS VD (point) (offset) [vertical distance]

| Parameter | Definition |
|---|---|
| point | The identifier of the new point that you want to create. You do not need to assign a point number if auto point numbering is on. |
| offset | The distance from the centerline to the new point. A positive distance indicates an offset to the right, and a negative distance indicates an offset to the left. |
| vertical distance | The elevation difference from the instrument to the prism. If the prism is higher than the instrument, then this is a positive value. |

# Intersections

You can use the intersection commands to locate intersection points between bearings, azimuths, lines, and arcs, and to calculate the square offset.

After you find an intersection, you can save the intersection point by specifying a new point identifier.

## Creating a Bearing/Bearing Intersection

You can find the point of intersection between two infinite lines from existing points.

After an intersection is found, you can save it by specifying a new point identifier.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate a Bearing/Bearing intersection in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Intersections menu ➤ Bearing/Bearing.

3   Enter the starting point number of the first bearing.

4   Enter the bearing from that point.

5   Enter the quadrant for the first point.

6   Enter an offset for the first points.

7   Enter the starting point number of the second bearing.

8   Enter the bearing from that point.

9   Enter the quadrant for the second point.

10   Enter the offset for the second point.

**To calculate a Bearing/Bearing intersection, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    BB [point 1] [bearing 1] [quadrant 1] [offset 1] [point 2] [bearing 2] [quadrant 2] [offset 2]

**Command Line Example**

NE 1 100 100

NE 2 200 200

*BB 1 66.6667 1 50 2 33.3333 2 50*

! INTERSECTION # 1 NORTH:100.000000 EAST:200.000000

An intersection is located between a bearing of N66.6667"E drawn from point 1, with an offset distance of 50 feet to the right and a bearing of S33.3333"E drawn from point 2 with, an offset distance of 50 to the right.
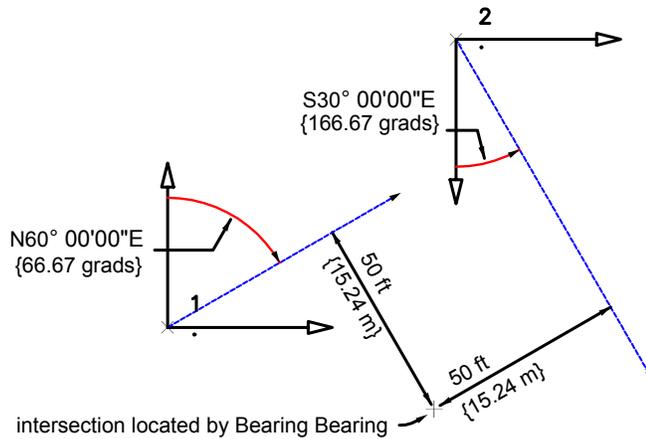
S30° 00'00"E
{166.67 grads}

N60° 00'00"E
{66.67 grads}

50 ft
{15.24 m}

50 ft
{15.24 m}

intersection located by Bearing Bearing

## Quick Reference

Survey Command Window Menu

Centerline ➤ Station/Offset

Survey Command Window Syntax

BB [point 1] [bearing 1] [quadrant 1] [offset 1] [point 2] [bearing 2] [quadrant 2] [offset 2]

| Parameter | Definition |
|---|---|
| point 1, point 2 | The existing point identifiers that establish the beginning of the bearings. |
| bearing 1, bearing 2 | The bearings for the lines from the existing points. Bearings establish a direction for each vector and are expressed in current angle units. |
| quadrant 1, quadrant 2 | The quadrants in which the bearings exist. The possible values are: 1 (for NE), 2 (for SE), 3 (for SW), and 4 (for NW). |
| offset 1, offset 2 | The offsets from the lines. This acts as if the lines are moved X feet or meters to the left or right. An offset to the right is a positive number, and an offset to the left is a negative number. If you do not want an offset, then use zero. |

# Creating an Azimuth/Azimuth Intersection

You can find the point of intersection between two infinite lines from existing points.

After an intersection is found, you can save it by specifying a new point number.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate an azimuth/azimuth intersection in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** Click Intersections menu ➤ Azimuth/Azimuth.

**3** Enter the starting point number of the first azimuth.

**4** Enter the azimuth from that point.

**5** Enter an offset.

**6** Enter the starting point number of the second azimuth.

**7** Enter the azimuth from that point.

**8** Enter an offset.

**To calculate an azimuth/azimuth intersection, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** At the Command line, enter:

AZAZ [point 1] [azimuth 1] [offset 1] [point 2] [azimuth 2] [offset 2]

**Command Line Example**

NE 1 100 100

NE 2 200 200

*AZAZ 1 50 10 2 183.3333 -20*

! INTERSECTION # 1 NORTH:186.991377 EAST:219.229524

An intersection is located between an azimuth of 50.000 drawn from point 1 with an offset distance of 10 to the right, and an azimuth of 183.3333 drawn from point 2 with an offset distance of 20 to the left.
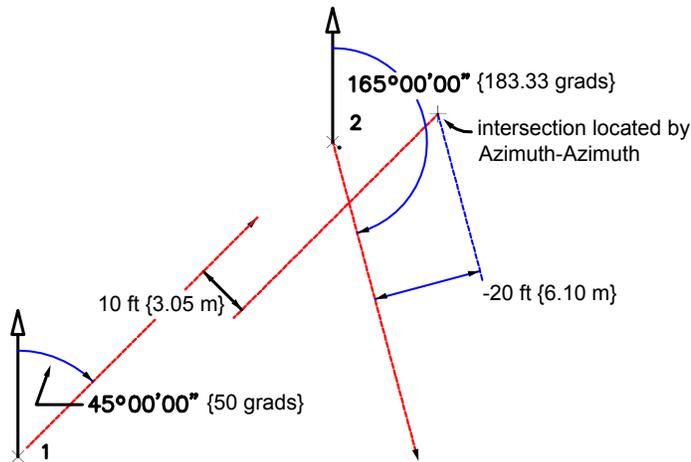


## Quick Reference

Survey Command Window Menu

Intersections ➤ Azimuth/Azimuth

Survey Command Window Syntax

AZAZ [point 1] [azimuth 1] [offset 1] [point 2] [azimuth 2] [offset 2]

| Parameter | Definition |
|---|---|
| point 1, point 2 | The existing point identifiers defining the beginning of the azimuths |
| azimuth 1, azimuth 2 | The azimuths for the lines from the existing points (point 1 and point 2). Azimuths establish a direction for each vector and are expressed in current angle units. |
| offset 1, offset 2 | The offsets from the lines. This acts as if the lines are moved X feet or meters to the left or right. An offset to the right is a positive number, and an offset to the left is a negative number. If you do not want an offset, then use zero. |

# Creating a Line/Line Intersection

You can find the point of intersection between two lines defined from existing points. After an intersection is found, you can save it by specifying a new point number.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a Line/Line intersection in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Intersections menu ➤ Line/Line.

3 Enter the starting point identifier of the first line.

4 Enter the second point identifier to define the first line.

5 Enter an offset.

6 Enter the starting point identifier of the second line.

7 Enter the second point identifier to define the second line.

8 Enter an offset.

**To create a Line/Line intersection, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:
LNLN [point 1] [point 2] [offset 1] [point 3] [point 4] [offset 2]

**Command Line Example**

NE 1 100 100

NE 2 200 200

NE 3 175 200

NE 4 100 200

*LNLN 1 2 100 3 4 -100*

! INTERSECTION # 1 NORTH:158.578644 EAST:300.000000

An intersection is located between a line drawn between points 1 and 2 with an offset of 100 feet to the right and a line drawn between points 3 and 4 with an offset of 100 feet to the left.
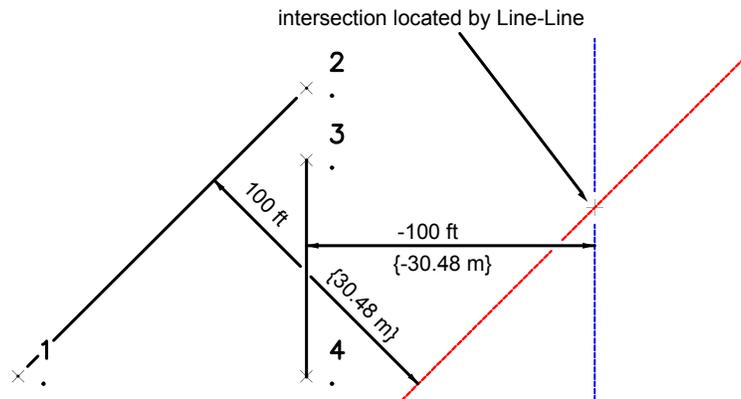


## Quick Reference

Survey Command Window Menu

Intersections ➤ Line/Line

Survey Command Window Syntax

LNLN [point 1] [point 2] [offset 1] [point 3] [point 4] [offset 2]

| Parameter | Definition |
| --- | --- |
| point 1, point 2 | The point identifiers that establish the position and direction of vector 1. Point 1 is the end point and Point 2 is the start point. These points can be any points including figure points. |
| offset 1, offset 2 | The offsets from the lines. This acts as if the vectors are moved X feet or meters to the left or right. An offset to the right is a positive number, and an offset to the left is a negative number. If you do not want an offset, then use a zero for the offset. |
| point 3, point 4 | The point numbers that establish the position and direction of vector 2. Point 3 is the end point and point 4 is the start point. These points can be any points including figure points. |

# Creating a Square Offset Intersection

You can determine the offset, distance, and point of intersection where a right angle would be projected onto an infinite line (defined by two points). After an intersection is found, you can save it by specifying a new point identifier.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To create a square offset in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2  Click Intersections menu ➤ Square Offset.

3  Enter the starting point identifier of the line.

4  Enter the second point identifier to define the line.

5  Enter the point identifier of the point offset from the line.

   AutoCAD Civil determines the offset distance of the point (entered in step 5) from the line established by the first two points. This establishes one known side distance and two known side directions. Using this information, AutoCAD Civil calculates the corners of a square and creates the corner of the square offset from the line (established in steps 3 and 4) at the same distance as defined by the point entered in step 5.

**To create a square offset, using the survey command language**

1  In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2  At the Command line, enter:

   SQ [reference 1] [reference 2] [reference]
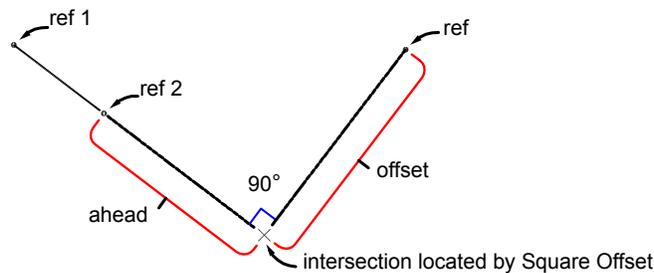


**Command Line Examples**

The following example shows a valid entry using the Square Offset command:

SQ 302 303 501

This example finds the point that creates a right angle between the line from point 302 to point 303 and point 501.

SQ 13 14 501

This example finds the point that creates a right angle between the line from point 13 to point 14 and point 501.

The Square Offset command lists the ahead distance and offset distance to the point of intersection.

## Quick Reference

Survey Command Window Menu

Intersections ➤ Square Offset

Survey Command Window Syntax

SQ [reference 1] [reference 2] [reference]

| Parameter | Definition |
| --- | --- |
| reference 1 | The existing end point on the vector. It can be any type of point, including a figure point. |
| reference 2 | The existing start point on the vector. Ref 1 and Ref 2 establish a direction on the vector. |
| reference | The point off the vector. The intersection point creates a right angle between the vector and this point. |

# Creating an Arc/Bearing Intersection

You can calculate the intersection between a line and an arc. The line is established by a point, bearing, and offset.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate the intersection of an arc and bearing line in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2  Click Intersections menu ➤ Arc/Bearing.

3  Enter the point identifier of the arc center.

4  Enter the radius of the arc.

5  Enter the starting point identifier of the line.

6  Enter the bearing of the line.

7  Enter the quadrant of the line.

8  Enter an offset.

9  Enter one of the following options:

■ N: To select the northern-most intersection.

■ S: To select the southern-most intersection.

■ E: To select the eastern-most intersection.

■ W: To select the western-most intersection.

■ R: To select the solution nearest.

■ F: To select the solution farthest.

■ 1: To select intersection 1.

■ 2: To select intersection 2.

■ A: To select all the intersections.

■ P: To pick the intersection with your pointing device

**To calculate the intersection of an arc and bearing line, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
    ARCB [point] [radius] [point 1] [bearing] [quadrant] [offset]

**Command Line Example**

NE 1 100 100

NE 2 200 200

*ARCB 2 200 1 50 1 -50*

! INTERSECTION # 1 NORTH:372.285978 EAST:301.575300

! INTERSECTION # 2 NORTH:98.424700 EAST:27.714022

Intersections are located between an arc radius of 200 with its center at point 2 and a bearing of 50.0000 drawn from point 1 with an offset distance of 50 to the left.
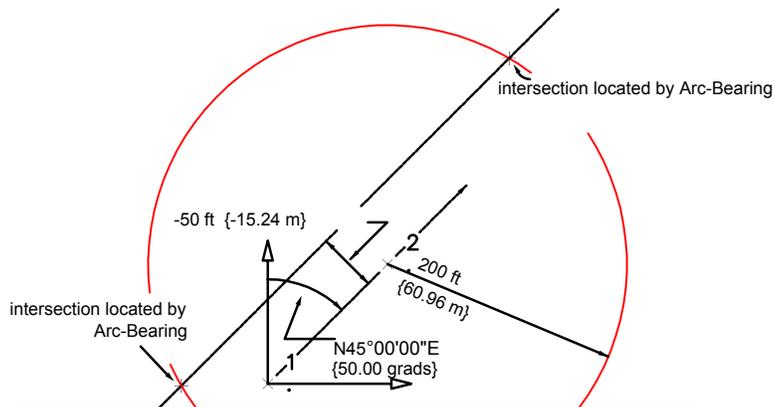


## Quick Reference

Survey Command Window Menu

  Intersections ➤ Arc/Bearing

Survey Command Window Syntax

  ARCB [point] [radius] [point 1] [bearing] [quadrant] [offset]

| Parameter | Definition |
|-----------|------------|
| point | The radial point. This is an existing point that is used as the center point for the arc. |
| radius | The radial distance. This is the distance in feet or meters from the radial point to the arc. |
| point 1 | The existing point from which a vector extends. It can be any type of point including a figure point. |

| Parameter | Definition |
|-----------|------------|
| bearing | The bearing of the line from the existing point. A bearing establishes a direction for the vector and is expressed in current angle units. |
| quadrant | The quadrant in which the bearing exists. The possible values are: 1 (NE), 2 (SE), 3 (SW), and 4 (NW). |
| offset | The offset left or right from the vector. This acts as if the line is moved X feet. An offset to the right is a positive number, and an offset to the left is a negative number. If you do not want an offset, then use zero for the offset. |

## Creating an Arc/Azimuth Intersection

You can find the points of intersection between an arc and a line that is established by a point, azimuth, and offset.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate the intersection of an arc and a line determined by an azimuth in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2  Click Intersection menu ➤ Arc/Azimuth.

3  Enter the point number of the arc center.

4  Enter the radius of the arc.

5  Enter the starting point number of the line.

6  Enter the azimuth of the line.

7  Enter an offset.

8  Enter one of the following options:

- N: To select the northern-most intersection.

- S: To select the southern-most intersection.

- E: To select the eastern-most intersection.

- W: To select the western-most intersection.

- R: To select the nearest intersection.

- F: To select the farthest intersection.

- 1: To select intersection 1.

- 2: To select intersection 2.

- A: To select all the intersections.

- P: To pick the intersection with your pointing device.

**To calculate the intersection of an arc and a line determined by an azimuth, using the survey command language**

**1** In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

**2** At the Command line, enter:

ARCAZ [point] [radius] [point 1] [azimuth] [offset]

**Command Line Example**

NE 1 100 100

NE 2 200 200

*ARCAZ 2 200 1 50 50*

! INTERSECTION # 1 NORTH:274.411634 EAST:385.641883

! INTERSECTION # 2 NORTH:30.099879 EAST:94.482471

Intersections are located between an arc radius of 200 with its center at point 2 and an azimuth of 50.0000 drawn from point 1 with an offset distance of 50 to the right.

**Intersection of arc and line determined by azimuth:**
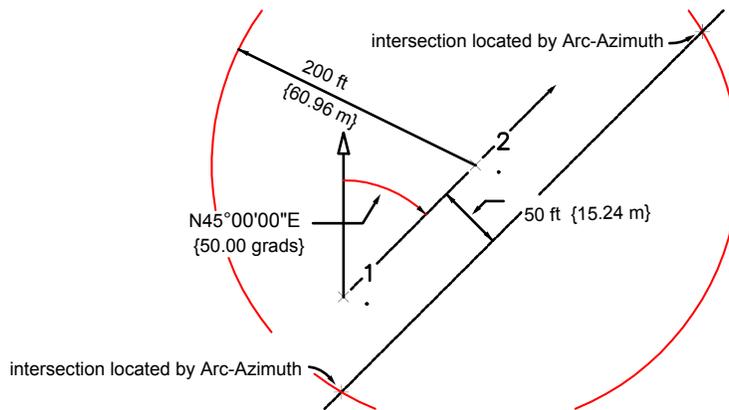


## Quick Reference

Survey Command Window Menu

Intersections ➤ Arc/Azimuth

Survey Command Window Syntax

ARCAZ [point] [radius] [point 1] [azimuth] [offset]

| Parameter | Definition |
|-----------|------------|
| point | The radial point. This is an existing point that is used as the center point for the arc. |
| radius | The radial distance in feet or meters for the first arc. Radial distance is the length of a line from the radius point to the arc. |

| Parameter | Definition |
|-----------|------------|
| point 1 | The existing point from which a vector extends. It can be any type of point including a figure point. |
| azimuth | The azimuth of the line from the existing point. An azimuth establishes the direction for the vector and is expressed as current angle units. |
| offset | The distance to offset the intersection. |

## Creating an Arc/Line Intersection

You can calculate the intersection(s) of a line established by two points, and an arc established by a radial point and radius.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate an Arc/Line intersection in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 Click Intersections menu ➤ Arc/Line.

3 Enter the point identifier of the arc center.

4 Enter the radius of the arc.

5 Enter the starting point identifier of the line.

6 Enter the ahead or the direction point for the line.

7 Enter an offset.

8 Enter one of the following options:
   ■ N: To select the northern-most intersection.

   ■ S: To select the southern-most intersection.

   ■ E: To select the eastern-most intersection.

   ■ W: To select the western-most intersection.

   ■ R: To select the nearest intersection.

   ■ F: To select the farthest intersection. 1: To select intersection 1.

   ■ 2: To select intersection 2.

   ■ A: To select all the intersections.

   ■ P: To pick the intersection with your pointing device.

**To calculate an Arc/Line intersection, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:
   ARCLN [point] [radius] [point 1] [point 2] [offset]

**Command Line Example**

NE 1 100 100

NE 2 200 200

NE 3 250 100

*ARCLN 3 200 1 2 -100*

! INTERSECTION # 1 NORTH:387.066972 EAST:245.645615

! INTERSECTION # 2 NORTH:104.354385 EAST:-37.066972

Intersections are located between an arc radius of 200 with the center at Point 3 and a line drawn between Points 1 and 2 with an offset distance of 100 to the left.

**Intersection of arc and line:**
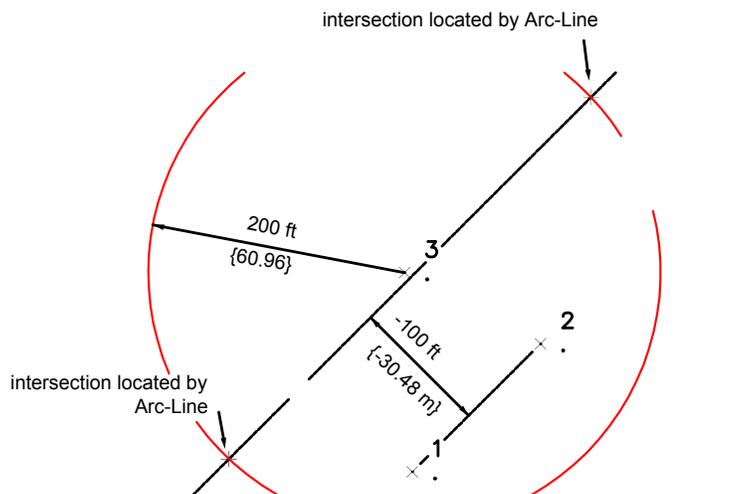


## Quick Reference

Survey Command Window Menu

  Intersections ➤ Arc/Line

Survey Command Window Syntax

  ARCLN [point] [radius] [point 1] [point 2] [offset]

| Parameter | Definition |
| --- | --- |
| point | The point number of the radial (center) point. |
| radius | The radial distance in feet or meters. The radial distance is the length of a line from the radius point to the arc. |
| point 1, point 2 | The existing points that establish the position and direction of the line. Point 1 is the start point and point 2 is the end point. These points can be any points including figure points. |

| Parameter | Definition |
|-----------|------------|
| offset | The offset from the vector. This acts as if the line is moved a specified number of feet (or meters) to the left or right. An offset to the right is a positive number, and an offset to the left is a negative number. If you do not want any offset, then use zero. |

# Creating an Arc/Arc Intersection

You can calculate the intersection(s) of two arcs, each defined by an existing center point and a radius.

**NOTE** For parameter descriptions, see the Quick Reference tab.

**To calculate the intersection of two arcs in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   Click Intersections menu ➤ Arc/Arc.

3   Enter the point identifier of the first arc center.

4   Enter the radius of the first arc.

5   Enter the point identifier of the second arc center.

6   Enter the radius of the second arc.

7   Enter one of the following options:

- ■ N: To select the northern-most intersection.

- ■ S: To select the southern-most intersection.

- ■ E: To select the eastern-most intersection.

- ■ W: To select the western-most intersection.

- ■ R: To select the nearest intersection.

- ■ F: To select the farthest intersection.

- ■ 1: To select intersection 1.

- ■ 2: To select intersection 2.

- ■ A: To select all the intersections.

- ■ P: To pick the intersection with your pointing device.

**To calculate the intersection of two arcs, using the survey command language**

1   In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2   At the Command line, enter:
ARCARC [point 1] [radius 1] [point 2] [radius 2]

**Command Line Example**

NE 1 100 100

NE 2 200 200

*ARCARC 1 100 2 200*

! INTERSECTION # 1 NORTH:8.856217 EAST:141.143783

! INTERSECTION # 2 NORTH:141.143783 EAST:8.856217

Intersections are located between an arc radius of 100 with a center at Point 1 and an arc radius of 200 with its center at Point 2.
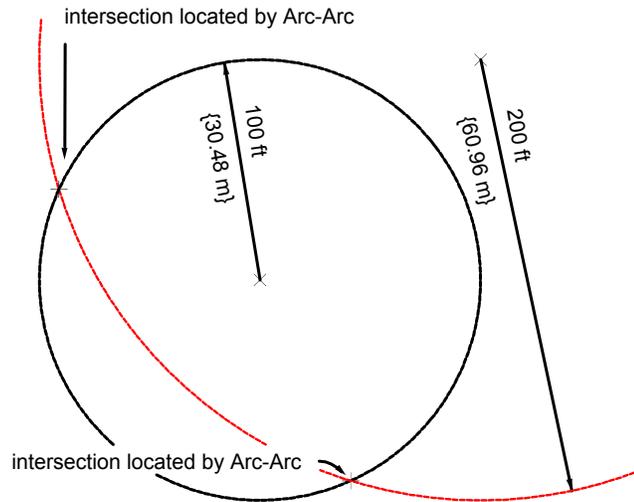


## Quick Reference

Survey Command Window Menu

 Intersections ➤ Arc/Arc

Survey Command Window Syntax

 ARCARC [point 1] [radius 1] [point 2] [radius 2]

| Parameter | Definition |
|-----------|------------|
| point 1 | The point number of radial (center) point 1. |
| radius 1 | The radial distance in feet or meters for the first arc. The radial distance is the length of a line from the radius point to the arc. |
| point 2 | The point number of radial (center) point 2. |
| radius 2 | The radial distance in feet or meters for the second arc. |

# Saving Intersection Points

After you find an intersection using the Intersection commands, you can save the new intersection point.

**To save the intersection point, using the survey command language**

1 In Toolspace, on the Survey tab, right-click the network to edit, and click Survey Command Window.

2 At the Command line, enter:

SAVE [NORTH, SOUTH, EAST, WEST, 1, 2, ALL] (point) (description)

---

**NOTE** For parameter descriptions, see the Quick Reference tab.

---

**Command Line Example**

SAVE NORTH 5

This saves the north intersection solution to point number 5.

## Quick Reference

Survey Command Window Syntax

SAVE [NORTH, SOUTH, EAST, WEST, 1, 2, ALL] (point) (description)

| Parameter | Definition |
| --- | --- |
| NORTH | Saves the northern most intersection. |
| SOUTH | Saves the southern most intersection. |
| EAST | Saves the eastern most intersection. |
| WEST | Saves the western most intersection. |
| 1 | Saves intersection 1. |
| 2 | Saves intersection 2. |
| ALL | Saves all intersections. |
| point | Saves the intersection nearest a point that you select. |
| description | An optional description associated with the point. If you use a description key, then specific information is assigned to the point. |

# Batch Files

Use batch files to record all survey commands that you use.

You can use the commands on the Batch file menu and the Output menu in the Survey Command Window to provide a historical record of a surveyor's decisions. A best practice is to create a new survey network to use in the Survey Command Window where you can perform calculations involving points from other existing survey networks. By using a new network, the data that is being input is captured in the batch file, where it can be edited and run with the corrections, and it will not affect the integrity of the other survey networks.

These commands are from data that is downloaded from a field book or entered from either the menus or at the Survey Command Window. You can open this file in your text editor to review and edit it.

After you make changes in the batch file, you can use the Run command to insert the data and draw the points in your drawing. The Run - Walk Through command downloads the data from the batch file and prompts you to press a key after each command in the batch file.

To use batch files, select the Use Batch File setting in the Survey Database Settings. For more information, see Database Settings (page 216).

## Setting the Batch File

Enable default batch file functionality in the Survey Database Settings.

You can also enable and change the batch file name in the Survey Command Window or by using the survey command language.

**To set the batch file**

1 In Toolspace, on the Survey tab, expand the Survey Databases collection, right-click the database, and click Edit Survey Database Settings.

2 Expand the Survey Command Window property group.

3 Ensure that Use Batch File is selected and enter the name for the batch file.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ right-click <database-name> ➤ Edit Survey Database Settings

Survey Command Window

Use Batch File
and
Batch File ➤ Set File Name

## Editing a Batch File

Use the Edit command to load the current batch file into your text editor, where you can view and edit it.

AutoCAD Civil uses Notepad as the default text editor. You can specify another editor in the Survey User Settings dialog box.

**NOTE** Some text editors format files. The batch file must be saved in standard ASCII format to work correctly.

**To edit a batch file**

1 In Toolspace, on the Survey tab, expand the Networks collection and right-click a network item. Click Survey Command Window.

2 In the Survey Command Window, click Batch File menu ➤ Edit.

3 The batch file is opened in a text editor.

### Quick Reference

Survey Command Window

Batch File ➤ Edit

# Running a Batch File

Use the Run command to load the batch file into the current project.

Running a batch file updates your drawing with any points and line work specified in the file (if the Insert settings are enabled). For information, see Setting Import Defaults (page 211).

The Run - Slow Motion command loads the batch file at a slower rate. You can watch each command entry as it is added to and displayed in your drawing.

Use the Run - Walk Through command to stop the action at every file line as it is read into the drawing. This command is very helpful when you are looking for problems in the data.

**To run a batch file**

1   In Toolspace, on the Survey tab, expand the Networks collection and right-click a network item. Click Survey Command Window.

2   In the Survey Command Window, click Batch File menu ➤ Run.

**To run a batch file in slow motion**

1   In Toolspace, on the Survey tab, expand the Networks collection and right-click a network item. Click Survey Command Window.

2   In the Survey Command Window, click Batch File menu ➤ Run - Slow Motion.

**To walk through a batch file**

1   In Toolspace, on the Survey tab, expand the Networks collection and right-click a network item. Click Survey Command Window.

2   In the Survey Command Window, click Batch File menu ➤ Run - Walk Through.

3   Press Enter after each line is displayed to continue. You must repeat this until you import the whole file.

### Quick Reference

Survey Command Window

Batch File ➤ Run
or
Batch File ➤ Run - Slow Motion
or
Batch File ➤ Run - Walk Through

# Survey Figures

# 12

You can use the Survey Toolspace and Survey Figure commands to create and edit survey figures, as well as to perform figure inquiries, identifying figures, and checking them for exact course and closure.

## Creating Figures

When you create a figure, it is stored in the survey database and may be also inserted into the active drawing depending on the Survey User Settings. Create a figure by either entering coordinates or referencing an existing point, and then giving a distance and direction to the next point.

Each figure consists of points or locations that have a relationship. You use figures to represent features such as fences, buildings, or roads. Figures are always referenced by name, and can consist of letters and numbers. For example: ROAD_1. One survey database may contain many figures. The AutoCAD Civil Survey feature has options to perform figure inquiries that identify a figure and determine exact courses and closure.

**NOTE**  Numeric figure names should not start with the number zero. For example, a figure name of 235 is acceptable but 0235 is not.

## Closing Figures

Close a figure by joining the last point of a figure to the first point of the same figure using the CLOSE command.

**NOTE**  For syntax descriptions, see the Quick Reference tab.

**To close a figure**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   CLOSE [figure]

**To close a building figure**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   CLOSE BUILDING

**To close a rectangle figure**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:

CLOSE RECTANGLE [offset].

The offset is the distance to the other side of the rectangle.

**Command Line Example of Closing a Figure**

BEG ROAD

NE SS 1000 1000

FIG BD 90 1 100

NE SS 1400 1400

CLOSE

END

**Command Line Example of Closing a Building Figure**

NE 1 1000 1000

AZ 1 2 90

STN 1

BS 2

BEG HOUSE

AD 101 90 100.21

AD 102 80.30 30.54

RT 80

RT 40

RT -40

CLOSE BLD

**Command Line Example of Closing a Rectangle Figure**

NE 10 2000 2000

STN 10

AZ 10 11 0

BS 11

BEG RECTA

AD 12 45 100

AD 13 30 50

CLOSE RECT -50

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

CLOSE
CLOSE BUILD
CLOSE RECTANG [offset]

Command Synonyms

CLOSE BLDG
CLOSE BLD
CLOSE RECT

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure you want to close. |
| offset | The distance to the other side of the rectangle when you are closing a rectangle figure. |

## Creating a Figure from an Object

Create a figure by selecting an object in the drawing.

When you create a figure, it is stored in the survey database and may be also inserted into the active drawing depending on the Survey User Settings.

You can create a survey figure from the following object types:

■ 2D Line

■ 3D Line

■ 2D Polyline

■ 3D Polyline

■ Feature Line

■ Lot Line

■ Parcel (select the parcel area label)

**To create a figure from an object**

1 In Toolspace, on the Survey tab, right-click the Figures collection ➤ Create Figure from Object.

2 Select an object in the drawing.

3 Specify settings in the Create Figure From Object dialog box.

4 Click OK. The figure is created and listed in the Figures collection.

**Quick Reference**

Ribbon

Click Home tab ➤ Create Design panel ➤ ▼ ➤ Create Figure From Object ⬚

Menu

Survey Menu ➤ Create Figure From Object ➤

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Figures ➤ right-click ➤ Create Figure From Object

Dialog Box

Create Figure From Object Dialog Box (page 2410)

## Creating Breaklines from Figures

Use figures that you located in your survey as surface breaklines.

In the process of making a surface model, you can easily select the figures to use as breaklines.

**To create breaklines from figures using Toolspace**

1  In Toolspace, on the Survey tab, right-click Figures ➤ Create Breaklines.

2  In the Create Breaklines dialog box, click the Select Surface drop-down list and click a surface name or click ⬚ to open the New Surface dialog box, where you can create a new surface. Figures that have the Breaklines fields set to Yes are displayed in the list view part of the dialog box.

3  Optionally, to prevent a breakline from being created from a figure, in the list view, click the Breakline field and set the value to No.

4  Click OK to create the breaklines. The selected breaklines are inserted into to the drawing, or if necessary updated if they already exist in the drawing.

5  In the Add Breaklines dialog box (page 2360), enter a Description for the breakline operation, specify the Type of breaklines, or specify the Mid-ordinate distance for any breaklines that have curved segments.

**Quick Reference**

Toolspace

Survey tab: Survey Databases ➤ <survey database name> ➤ Figures collection ➤ right-click Create Breaklines

## Inserting Figures into Drawings

Insert survey figures into drawings.

**To insert a survey figure into a drawing**

➤  In Toolspace, on the Survey tab, expand the Figures collection and right-click a <figure name> ➤ Insert Into Drawing.

### Quick Reference

Toolspace

Survey tab: Figurescollection ➤ right-click <named figure> ➤ Insert Into Drawing

# Creating Points on a Figure

Add survey points to the vertices of an existing figure by selecting a figure and then setting a survey point at each figure vertex.

**To create points on a figure in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   Click Figures menu ➤ Create Points on Figures.

3   Select the figure that you want to create the points on.

4   At the Command line, enter the starting point number or accept the default. A survey point is created for each figure vertex.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <network name> ➤ Survey Command Window

# Creating a Point of Curvature on a Figure

Create a curve (on a figure) with three shots: the first at the beginning, the second on the arc between the beginning and end of the curve, and the third at the end of the curve.

**To draw a 3-point curve**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the Command line, enter:
    C3

**Command Line Example**

BEG CURVE

C3

AD VA 11 85 200 90

AD VA 12 82 200 90

AD VA 13 80 190 90

END

The following illustrates the Three Point Curve command:
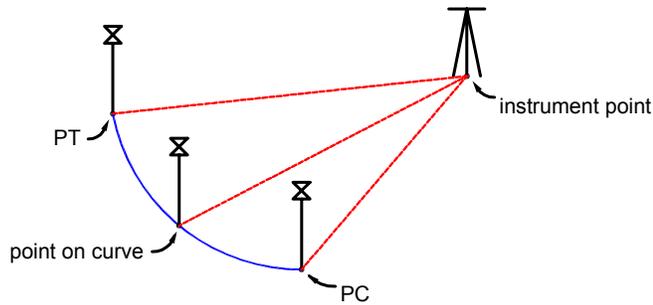
### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

C3

## Creating Multi-point Curves in a Figure

Within an active Figure, where (BEG <Figure name>) has been invoked, you can enter a figure command to specify subsequent points to define multiple points on a curve.

After the points on the curve have been defined, you can enter a command to end the multi-point curve and subsequent points will define line segments. The figure object created from the multi-point curve contains contiguous arc segments for each multi-point curve range.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To create a multi-point curve in a figure**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
  BEGIN [figure]

3 With the BEGIN command active, enter:
  MCS

  The next points you enter will define the points on the curve.

4 Enter MCE to end the curve on the figure. Entering MCE does not end the figure. Enter END to end the active figure or enter BEGIN to start a new figure.

**Command Line Example**

NE 1000 1000

BEG EP1

PT1

NE SS 2 1100 1100

MCS

NE SS 3 1140 1120

NE SS 4 1800 1110

NE SS 5 1180 1070

MCE

NE SS 6 1160 1020

END

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

BEGIN [figure] and then MCS

Command Synonyms

BEG and MCS to start curve
MCE to end the Multi-point Curve command

| Parameter | Definition |
|-----------|------------|
| figure | The name of the figure. |

# Creating a Curve on a Figure Using Point of Curvature

Specify that the next shot is a Point of Curvature (PC) in the current figure. The shot immediately following the next shot is on the Point of Tangency (PT). This command creates a tangential arc from the last leg in the figure.

**To specify that the next shot is a Point of Curvature (PC)**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the Command line, enter:
    PC

**Command Line Example**

PC

AD 13 30 25 ! Shot on PC

AD 14 33 24 ! Shot on PT

END

The following illustrates the Point of Curvature command:
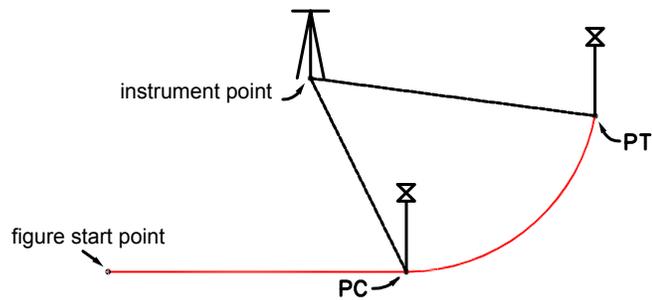
## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

PC

# Creating a Curve in the Current Figure

Create a curve in the current figure using two known variables.

**To create a curve**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  In the command window, enter any capitalized curve parameters, plus the radius and the curve parameter value:

- CURVE DELTA [radius] [value]

- CURVE LENGTH

- CURVE DEFL

- CURVE TANGENT

- CURVE CHORD

- CURVE MID

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

CURVE DELTA [radius] [value]
CURVE LENGTH [radius] [value]

CURVE DEFL [radius] [value]
CURVE TANGENT [radius] [value]
CURVE CHORD [radius] [value]
CURVE MID [radius] [value]

Command Synonyms

CURVE TAN

| Parameter | Definition |
|---|---|
| DELTA | The value parameter indicates the angle of the circular curve. |
| LENGTH | The value parameter indicates the length of the circular curve. |
| DEFL | The value parameter indicates the deflection angle between the current figure direction and the chord. |
| TANGENT, TAN | The value parameter indicates the tangent distance of the curve. |
| CHORD | The value parameter indicates the chord length of the curve. |
| MID | The value parameter indicates the mid-ordinate distance of the desired curve. |
| radius | The distance from a center point to the curve. You must enter this value in the current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| value | A curve value that depends on the curve parameter that you specified. For example, if you specify CHORD, then the value is the chord length. You must enter all distances in the current survey database distance units, and angles in the current survey database angular units, either DMS, grads, decimal degrees, mils, or radians. If you enter a negative value, then the negative sign is ignored. The radius component described above is used to control the direction of the curve. |

## Creating a Figure Vertex Using an Angle Distance

Create a figure vertex using an angle turned from a previous backsight. Two or more vertices must exist in the figure to use this command.

**To create a figure vertex using an angle and distance**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   FIGURE ANG-DIST [angle] [distance]

**Command Line Example**

FIG NE 1000 1000

FIG ZD 0 222.2222

FIG AD 55.5556 304.80

AZIMUTH: 255.5556 DISTANCE: 304.8000

NORTH: 1026.301 EAST: 766.510

FIG AD 83.3333 15.240

AZIMUTH: 138.8889 DISTANCE: 15.2400

NORTH: 1017.559 EAST: 778.993

END

---

**NOTE** Distances are in the current survey database units, unless you enter a qualifying suffix.

---

The following illustrates the Figure Angle Distance command:
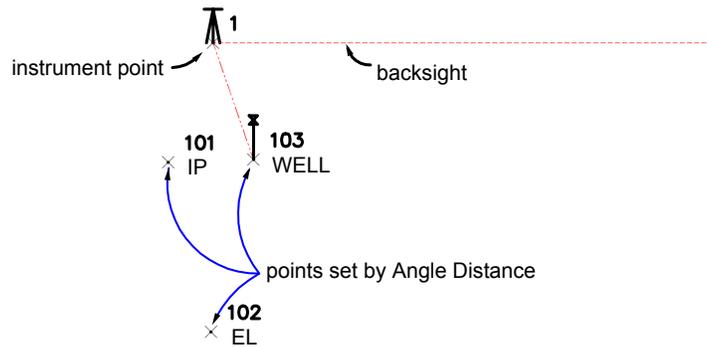


## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

FIGURE ANG-DIST [angle] [distance]

Command Synonyms

FIG AD

| Parameter | Definition |
|-----------|------------|
| angle | The clockwise angle to turn from the last vertex in the figure. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, radians, or mils. |
| distance | The distance from the current vertex in the figure. Enter this value in the current survey database distance units. |

# Creating a Figure Using an Azimuth Distance

Create a new vertex from an existing vertex in the figure using an azimuth and distance. At least one vertex must exist in the figure for this command to work.

**To create a new vertex**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   FIGURE AZ-DIST [azimuth] [distance]

**Command Line Example**

BEGIN 1_95

FIG NE 2000 2000

FIG ZD 50.5556 91.440

AZIMUTH: 50.5556 DISTANCE: 91.4400

NORTH: 2057.6291 EAST: 2070.9941

CRV DELTA 152.400 9.1440

CRV LENGTH -243.840 152.400

END

The following illustrates the Azimuth Distance command:
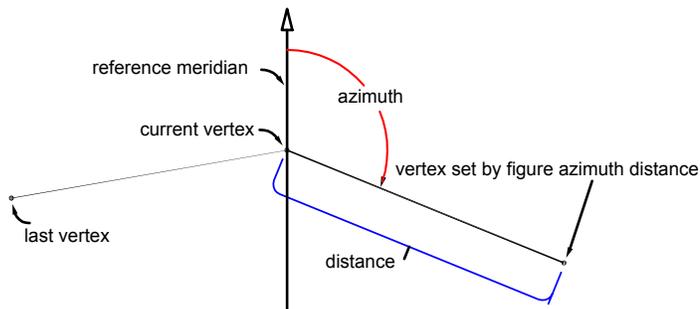


## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

FIGURE AZ-DIST [azimuth] [distance]

Command Synonyms

FIG ZD

| Parameter | Definition |
| --- | --- |
| azimuth | The desired azimuth from the current vertex to the new vertex. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, radians, or mils. |
| distance | The distance from the current vertex in the figure to the new vertex. Enter this value in the current survey database distance units. |

## Creating a Figure Vertex Using a Deflection Distance

Create a figure vertex using a deflection angle and distance. The deflection angle is turned clockwise from the current azimuth. Two or more vertices must exist in the figure to use this command.

**To create a figure vertex using a deflection angle and distance**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
    FIGURE DEFL-DIST [angle] [distance]

**Command Line Example**

NE 20 1200 1200

NE 21 1400 1300

BEGIN STONE_ST

PT 20

PT 21

FIG DD -33.3333 91.440

RT -91.440

FIG ZD (AZ 21 20) 60.960

CLOSE

The following illustrates the Figure Deflection Distance command:
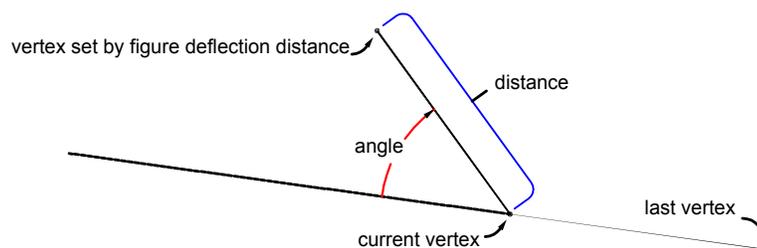
### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

FIGURE DEFL-DIST [angle] [distance]

Command Synonyms

FIG DD

| Parameter | Definition |
|-----------|------------|
| angle | The deflection angle as measured to the new vertex.Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, radians, or mils. |
| distance | The distance from the current vertex in the figure. Enter this value in the current survey database distance units. |

## Creating a Figure Vertex Using a Bearing Distance

Create a new vertex in a specified direction from the current figure vertex. At least one vertex must exist in the figure for this command to work properly.

**To create a new vertex in a direction from the current figure**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
FIGURE BRG-DIST [bearing] [quadrant] [distance]

**Command Line Example**

BEG BFIG

FIG NE 1000 1000

FIG BD 61.1111 1 30.480

AZIMUTH: 61.1111 DISTANCE: 30.4800

NORTH: 1014.6902 EAST: 1026.7063

END

The following illustrates the Figure Bearing Distance command:
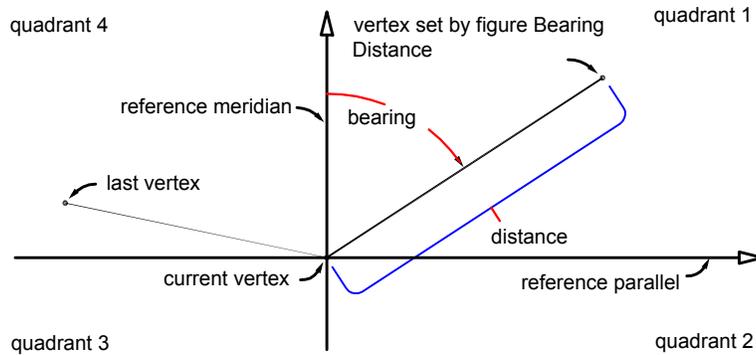
## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

FIGURE BRG-DIST [bearing] [quadrant] [distance]

Command Synonyms

FIG BD

| Parameter | Definition |
|-----------|------------|
| bearing | The desired bearing. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, mils, or radians. |
| quadrant | Specify the quadrant using one of the following numbers: 1 (for NE), 2 (for SE), 3 (for SW), or 4 (for NW). |
| distance | The measured distance from the current vertex to the new vertex. Enter this value in the current survey database distance units. |

## Creating a New Vertex with Known Coordinates

Create a new vertex by using known coordinates.

**To create a new figure vertex using known coordinates**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
FIGURE NORTH-EAST [northing] [easting]

**Command Line Example**

BEG CITY_LIMIT

FIG NE 2000 2000

FIG NE 1400 1400

FIG ZD 290 4000

FIG ZD 45 3000

CLOSE

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

FIGURE NORTH-EAST [northing] [easting]

Command Synonyms

FIG NE

| Parameter | Definition |
|-----------|------------|
| northing | The northing of the new vertex location that you want to create. |
| easting | The easting of the new vertex location that you want to create. |

# Creating a Figure Vertex Using an Existing Point as the Next Point

Enter an existing point identifier to use as the next vertex location in the figure you are drawing.

**To specify an existing survey point in the survey database to be the next location of the figure**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   POINT [point]

**Command Line Example**

NE 1 1000 1000

BEG FIG

FIG NE 2000 2000

PT 1

BEARING: S 45-00-00 W DISTANCE: 1414.2136

NORTH: 1000.0000 EAST: 1000.0000

END

### Quick Reference

Toolspace

> Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

> POINT [point]

Command Synonyms

> PNT
> RECALL
> PT

| Parameter | Definition |
| --- | --- |
| point | The point identifier of the point to use in the figure. This point must already exist in the survey database. |

## Creating Right Angle Segments on a Figure

Define left and right turns, creating square sides on your figure. This is useful when drafting building footprints. At least two vertices must exist in the current figure for this command to work properly.

**To define a right turn**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   RT-TURN [distance]

**Command Line Example**

BEG HOUSE

NE 1 1000 1000

FIG ZD 100 30.480

RT 30.480

AZIMUTH: 190-00-00 DISTANCE: 30.4800

NORTH: 964.6903 EAST: 1024.7241

RT -3.048

AZIMUTH: 100-00-00 DISTANCE: 3.0480

NORTH: 964.1610 EAST: 1027.7258RT 15.240

AZIMUTH: 190-00-00 DISTANCE: 15.2400

NORTH: 949.1525 EAST: 1025.0794

CLOSE BLD

END

The following illustrates the Right Turn command:
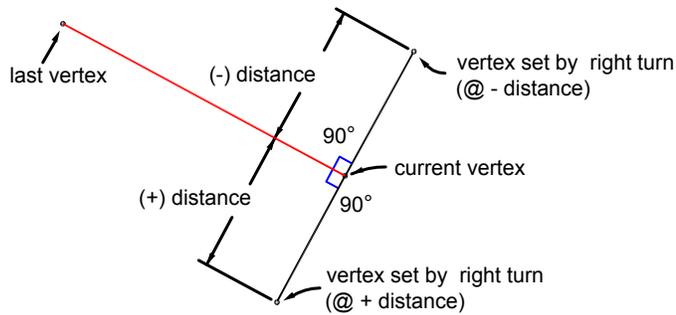
## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

RT-TURN [distance]

Command Synonyms

RIGHT
RT

| Parameter | Definition |
|-----------|------------|
| distance | The distance from the current vertex to the new vertex. Enter this value in the current survey database distance units. |

# Creating a Point at the Current Location

Create a point at the current location as you create a figure.

**To create a point at the current location in a figure**

1. In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2. At the Command line, enter:
   SET (point)

**Command Line Example**

BEG FIG

FIG NE 1000 1000

FIG ZD 0 30.480

RT 60.960

FIG AD 50 60.960

SET 5

POINT 5 NORTH: 1077.1781 EAST: 1021.7757 ELEV: <Null>

END

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

SET (point)

| Parameter | Definition |
| --- | --- |
| point | The assigned point identifier. You do not need to assign a point number if auto point numbering is on. |

# Creating Non-Tangential Curves in Figures

You can use the following commands to create non-tangential curves in the Survey Command Window.

## Creating a Non-Tangential Curve in the Current Figure

Create a non-tangential curve in the current figure.

**To create a non-tangential curve**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   XC AD BULB [radius] [chord angle] [chord distance]

   **NOTE** Use the BULB parameter if the resulting curve is greater than 180 degrees.

**Command Line Example**

NE 1 1000 1000

NE 2 1000 1200

BEG ROADCL

PT 1

PT 2

XC ZD 100 45 120

SET 3

FIG DD 0 100

SET 4

XC AD 100 45 150

SET 5

END

The following illustrates a non-tangential curve:
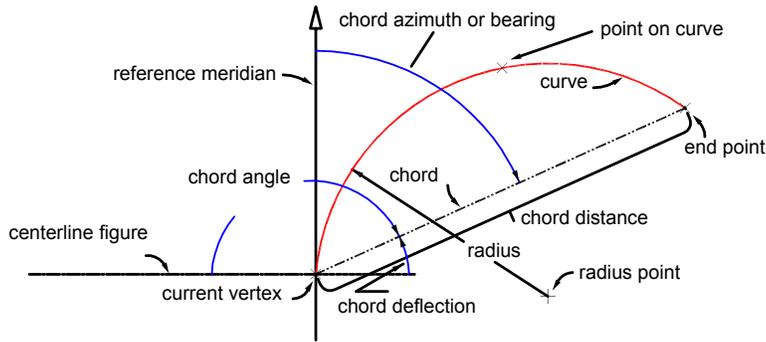


## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

XC AD BULB [radius] [chord angle] [chord distance]

| Parameter | Definition |
| --- | --- |
| radius | The distance from a center point to the curve. Enter this value in current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| chord angle | The clockwise angle to turn from the last vertex in the figure. |
| chord distance | The distance from the current vertex in the figure. |

# Creating a Curve on a Figure Using a Point on the Curve

Create a non-tangent curve on a figure using three points.

---
**NOTE** For syntax descriptions, see the Quick Reference tab.

---

**To create a curve on a figure using a point on the curve**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
   XC C3 [point on curve] [end point]

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

XC C3 [point on curve] [end point]

| Parameter | Definition |
|---|---|
| point on curve | The point identifier of an existing point on the curve. |
| end point | The point identifier of the end point of the curve. |

## Creating a Curve Using a Radius Point

Create a non-tangential curve using a radius point and point at the end of the curve.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To create a non-tangential curve using a radius point and a point at the end of the curve**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   XC PTS [radius] [radius point] [end point]

**Command Line Example**

NE 1 1000 1000

NE 2 1000 1200

NE 3 1084.8528 1284.8528

NE 6 985.8579 1298.9949

BEG ROADCL

PT 1

PT 2

XC PTS 100 6 3

FIG DD 0 100

SET 4

XC AD 100 45 150

SET 5

END

The following illustrates two non-tangential curves:
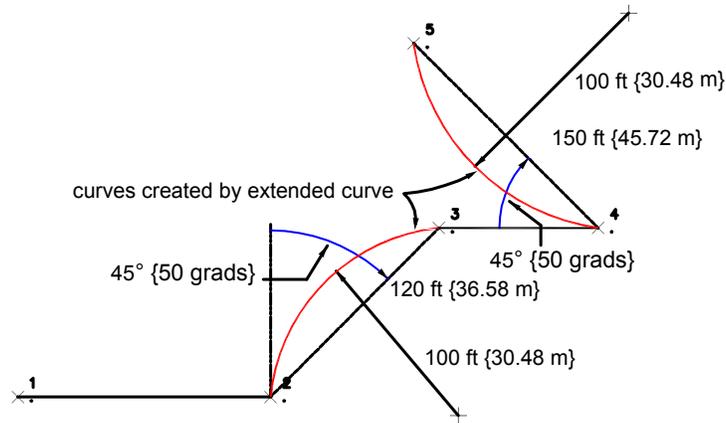
## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

XC PTS [radius] [radius point] [end point]

| Parameter | Definition |
|---|---|
| radius | The distance from a center point to the curve. Enter this value in current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| radius point | The point identifier of an existing point that is used as the radial point for the arc. |
| end point | The point identifier of the end point of the curve. |

# Creating a Curve Using the Chord Bearing

Create a non-tangential curve using the chord bearing.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To create a non-tangential curve using the chord bearing**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the Command line, enter:
    XC BD BULB [radius] [chord bearing] [quadrant] [chord distance]

### Quick Reference

Survey Command Window Syntax

XC BD BULB [radius] [chord bearing] [quadrant] [chord distance]

| Parameter | Definition |
| --- | --- |
| BULB | Creates a curve that is greater than 180°. |
| radius | The distance from a center point to the curve. Enter this value in the current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| chord bearing | The desired bearing. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, mils, or radians. |
| quadrant | Specify the quadrant using one of the following numbers: 1 (for NE), 2 (for SE), 3 (for SW), or 4 (for NW). |
| chord distance | The distance from the current vertex in the figure. Enter this value in the current survey database distance units. |

# Creating a Curve Using the Chord Deflection

Create a non-tangential curve using the chord deflection.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To create a non-tangential curve using the chord deflection**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the Command line, enter:
    XC DD BULB [radius] [chord deflection] [chord distance]

### Quick Reference

Survey Command Window Syntax

XC ZD BULB [radius] [chord deflection] [chord distance]

| Parameter | Definition |
| --- | --- |
| BULB | Creates a curve that is greater than 180°. |

| Parameter | Definition |
|---|---|
| radius | The distance from a center point to the curve. Enter this value in the current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| chord bearing | The desired bearing. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, mils, or radians. |
| chord distance | The distance from the current vertex in the figure. Enter this value in the current survey database distance units. |

## Creating a Curve Using the Chord Azimuth

Create a non-tangential curve using the chord azimuth.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To create a non-tangential curve using the chord azimuth**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:

XC ZD BULB [radius] [chord azimuth] [chord distance]

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

XC ZD BULB [radius] [chord azimuth] [chord distance]

| Parameter | Definition |
|---|---|
| BULB | Creates a curve that is greater than 180°. |
| radius | The distance from a center point to the curve. Enter this value in the current survey database distance units. Use a negative radius to draw the curve in a counter-clockwise (left) direction. |
| chord bearing | The desired bearing. Enter this value in the current survey database angular units, either DMS, grads, decimal degrees, mils, or radians. |
| chord distance | The distance from the current vertex in the figure. Enter this value in the current survey database distance units. |

# Modifying Figures

Use the Modify Figure commands to add points, segments and vertices to existing figures.

# Beginning a New Figure

You can create a figure by either entering coordinates or referencing an existing point, and then giving a distance and direction to the next point.

Within an active Figure where (BEG <Figure name>) has been invoked, you can enter a figure command to specify that subsequent points define multiple points on a curve. After the points on the curve have been defined, you can enter a command to end the multi-point curve and subsequent points will define line segments. The Polyline object created from the multi-point curve contains contiguous arc segments for each multi-point curve range.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To begin a new figure**

1  In Toolspace, on the Survey tab, right-click a Figure collection ➤ Modify Figure ➤ Begin New Figure.

2  At the Command line, enter a figure name.

3  Pick the first point. You can pick a point on the screen with object snaps.

4  Select the next vertex by selecting a point on the screen, or entering one of the following options:

   ■  AD and then the angle and distance.

   ■  BD and then the bearing and distance.

   ■  ZD and then the azimuth and distance.

   ■  DD and then the deflection distance.

   ■  RT and then a negative or positive right distance, which is a right angle departure from the last leg of the figure.

   ■  CU to create a curve, type the radius, and select one of the following options for creating the curve: Select entry (Tan/Chord/Delta/Ext/Mid/Length)<Length>:

   ■  PT and then the point number.

   ■  CL to close the figure, linking the last vertex with the first point.

**Command Line Example**

1000 1000

BEG EP1

PT 1

NE 2 1100 1100

MCS

NE 3 1140 1120

NE 4 1800 1110

NE 5 1180 1070

MCE

NE 6 1160 1020

END

When you use the Begin command to start another figure, any figure currently being drawn is automatically ended.

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Modify Figure ➤ Begin New Figure

Command Line Syntax

BEGIN [figure]

Command Synonyms

BEG

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure. |

# Extending a Figure from the Start Point

Add segments to the start point of an existing figure. The start point of the figure becomes the first point of the new segments.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To extend a figure from the start point in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click a Figure collection or in individual figure ➤ Modify Figure ➤ Start Figure.

2 Enter the figure name.

3 Define the next vertex by selecting the points on screen or by entering one of the following options:

■ AD and then the angle and distance

■ BD and then the bearing and distance.

■ ZD and then the azimuth and distance.

■ DD and then the deflection distance.

■ RT and then a negative or positive right distance, which is a right angle departure from the last leg of the figure.

■ CU to create a curve, type the radius, and select one of the following options for creating the curve: Select entry (Tan/Chord/Delta/Ext/Mid/Length)<Length>:

■ PT and then the point identifier.

■ CL to close the figure, linking the last vertex with the first point.

**Command Line Example**

AP ON 1

BEG ROAD

NE SS 1000 1000

NE SS 1200 1200

NE SS 1400 1400

BEG EP

FIG NE 1300 1300

FIG ZD 180 200

START ROAD

This example starts figure ROAD, which proceeds from N 1000 E 1000 with a referenced azimuth away from N 1300 E 1300.

**NOTE** The Start command, like the Continue and Begin commands, ends the current figure.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Modify FigureStart Figure

Command Line Syntax

START [figure]

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure that you want to extend. |

## Continuing a Figure

Add additional vertices to an existing figure using the last vertex entered as the starting point.

**NOTE** For syntax descriptions, see the Quick Reference tab.

**To continue a figure in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click a Figure collection or in individual figure ➤ Modify Figure ➤ Continue Figure.

2   Enter the figure name.

3   Define the next vertex by selecting the points on screen or by entering one of the following options:

   ■ AD and then the angle and distance

   ■ BD and then the bearing and distance.

   ■ ZD and then the azimuth and distance.

   ■ DD and then the deflection distance.

   ■ RT and then a negative or positive right distance, which is a right angle departure from the last leg of the figure.

   ■ CU to create a curve, type the radius, and select one of the following options for creating the curve: Select entry (Tan/Chord/Delta/Ext/Mid/Length)<Length>:

- PT and then the point identifier.

- CL to close the figure, linking the last vertex with the first point.

**Command Line Example**

NE 1 1000 1000

NE 2 1000 2000

STN 1

BS 2

BEG CURBA

AD 4 50 60

BEG CURBB

AD 5 60 80

AD 6 90 40

CONT CURBA

!Figure CURBA has been continued.

AD 7 20 60

END

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Modify FigureContinue Figure

Command Line Syntax

Continue [figure]

Command Synonyms

CONT

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure that you want to continue. |

# Editing Figures

Adjust the point data in a defined figure. After you enter the adjusted data, you can update and rename the figure.

The Figures Editor, which is accessible in the Survey Toolspace, provides a centralized way of editing and managing figure data.

**IMPORTANT** Figures that are created using a Linework Code Set do not automatically update when you modify survey data by updating figures, changing survey point descriptions, editing point coordinates through changes in observations, or network and traverse adjustment. In order to obtain the most accurate results, it is recommended that you process the linework each time you make modifications to the survey data. For more information, see Processing Survey Linework (page 251)

## Editing Figures from the Toolspace

Use the Survey Toolspace to view and edit figure data for either multiple figures or an individual figure.

The Figures Editor provides the same editing functionality as the Figure properties list view does in Toolspace, but is more suited for horizontal viewing of a tabulation of this type of data. The benefit of editing figures in the Figures Editor is that you can switch to the Survey Toolspace browse through the data, and then return to the Figures Editor and continue with your edits.

**NOTE** You can use the Survey Database Settings dialog box to set the Direction Type, Angle Unit, Distance/Coordinate Unit that is displayed for the Figures Editor vista.

**To view and edit figures**

1   In Toolspace, on the Survey tab, do one of the following:

   ■  To view and edit all figures, right-click the Figures collection ➤ Edit to display the Figures Editor.

   ■  To view and edit a single figure, expand the Figures collection and click the individual figure. Edit the figure data in the Toolspace properties list view.

2   Modify the fields as required by selecting the field and entering or selecting the new value.

   In the Breakline field, select the check box to indicate that the figure will be used as a surface breakline when you run the Create Breakline command.

   For more information, see Creating Breaklines from Figures (page 324). When you make a change to any figure data, the entire row is highlighted in bold until the change is saved to the survey database.

3   To edit the values in multiple rows, select the rows, right-click the column heading, click Edit. Enter the new value. The selected row values for the column are updated with the new value.

   **NOTE**  If no rows are selected, then all rows are updated.

4   To delete figures, click the figure(s) to delete, right-click, and click Delete.

   The figure icon 🖫 is replaced with ❌ and the row is set to read-only.

5   To undo the changes for a single row or multiple rows, select the row(s), right-click, and click Reload or Undelete. To undo all the changes in the vista, click 🔄 .

6   To apply the changes for a single row or multiple rows, select the row(s), right-click, and click Apply. To apply all changes in the vista, click 💾 .

7   Click ✅ to close the vista.

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Figures Collection

# Browsing to a Figure

Browse to a figure in the Figures collection on the Survey tab in Toolspace.

The figure is highlighted in the Figures collection on the Survey tab in Toolspace.

**To browse to a figure**

1  Click Survey tab ➤ Modify panel ➤ Browse To Survey Data Drop-down ➤ ⬛ Browse To Figure.

2  Select a figure in the drawing. The figure you select will be highlighted in the Figures collection on the Survey tab in Toolspace.

### Quick Reference

Ribbon

Survey tab ➤ Modify panel ➤ Browse To Survey Data Drop-down ➤ ⬛ Browse To Figure

Menu

Click Survey menu ➤ Browse ➤ Survey Figure

Command Line

BrowseToSvFigure

# Updating Survey Figure Objects

**To update the survey figure drawing object**

➤  Click Survey tab ➤ Modify panel ➤ Update Figure drop-down ➤ Update Figure From Survey Data ⬛ .

### Quick Reference

Ribbon

Click Survey tab ➤ Modify panel ➤ Update Figure drop-down ➤ Update Figure From Survey Data ⬛

Toolspace

Prospector tab ➤ Survey ➤ Figures ➤ <named figure> ➤ Update from Survey Data

Command Line

UpdateFromSvFigure

# Updating Survey Figure Data

**To update the survey database with changes made to the figure drawing object**

➤  Click Survey tab ➤ Modify panel ➤ Update Figure drop-down ➤ Update Survey Data From Drawing ⬛ .

**Quick Reference**

Ribbon

Click Survey tab ➤ Modify panel ➤ Update Figure drop-down ➤ Update Survey Data From Drawing 🖳

Toolspace

Survey tab ➤ Figures collection ➤ <named figure> ➤ Update from Survey Data

Command Line

UpdateSvFigureFromDwg

# Changing the Name of an Existing Figure

You can change the name of a figure.

**To change the name of a figure**

1 In Toolspace, on the Survey tab, do one of the following:
   - Right-click the Figures collection and click Edit to display the Figures Editor.
   - Click the figure to display its properties in the list view.

2 Do one of the following:
   - If you are working in the Figures Editor, select the figure you want to rename.
   - If you are working in the list view, click the figure name field.

3 Enter a new name for the figure.

**Quick Reference**

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Figures Collection

# Redrawing Adjusted Figures

You can redraw all the defined figures in your survey database based on the current point coordinates.

After you adjust and update survey points, you can update the figures based on adjusted coordinates.

**To update figures**

➤ In Toolspace, on the Survey tab, do one of the following:
   - To update all figures, right-click the Figures collection ➤ Update Figures.
   - To update a single figure, expand the Figures collection and right- click the individual figure ➤ Update Figure.

**Quick Reference**

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Figures collection ➤ Update Figures

# Offsetting a Figure

You can offset an existing figure by a specified distance. A positive value indicates an offset to the right; a negative value indicates an offset to the left.

**To offset a figure**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:

OFFSET [figure] [distance]

---

**NOTE** Numeric figure names should not start with the number zero. For example, a figure name of 235 is acceptable but 0235 is not.

---

**Command Line Example**

NE 30 2000 2000

NE 31 2500 2500

BEGIN EASTERN_AVE

PT 30

PT 31

CRV LENGTH 60.960 91.440

FIG DD 0 91.440

CRV DELTA -121.920 6.096

END

OFFSET EASTERN_AVE 15.240

OFFSET EASTERN_AVE -6.096

The following illustrates the Offset Figure command:
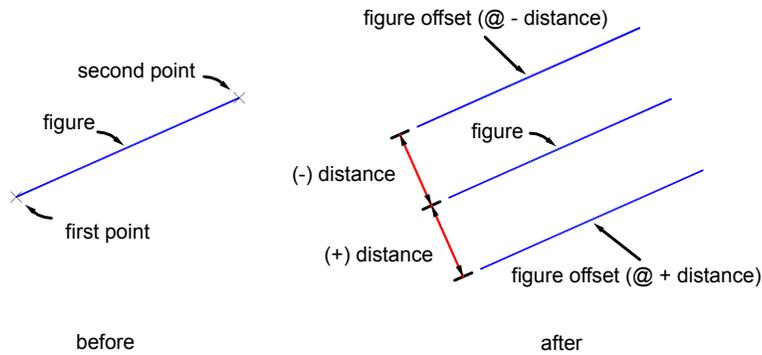
second point

figure

first point

before

figure offset (@ - distance)

figure

(-) distance

(+) distance

figure offset (@ + distance)

after

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

OFFSET [figure] [distance]

Command Synonyms

OFFSETS

| Parameter | Definition |
|-----------|------------|
| figure | The name of the figure you want to extend. |
| distance | The offset distance from the current point. Enter this value in the current survey database distance units. |

# Ending a Figure

Use the End command to disable the DRAW mode. After you disable the DRAW mode, you can use commands not related to the figure, such as commands to add points or plot the drawing.

**NOTE** The Begin and Continue commands also end the current figure. However, these commands also cause a different figure to become the current figure.

**To indicate the end of a figure**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the Command line, enter:
   END

**Command Line Example**

NE 20 1200 1200

NE 21 1400 1300

BEG CONYROAD

PT 20

PT 21

CRV DELTA 300 90

FIG DD 0 300

END

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

END

Command Synonyms

E

# Deleting Figures

Delete figures from either the Toolspace list view or the Figure Editor vista, and also in the Survey Command Window.

Use the Figures Editor vista to mark figures for deletion and then accept or roll back the deletions.

**To delete figures from the Toolspace tree**

1   In Toolspace, on the Survey tab, expand the Figures collection to display the individual figures.

2   Right-click the figure and click Delete.

**To delete figures from the Toolspace list view**

1   In Toolspace, on the Survey tab, click the Figures collection to display the Figures list view.

2   Right-click the figure in the list view, and click Delete.

**To delete figures from the Figures Editor**

1   In Toolspace, on the Survey tab, right-click the Figures collection ➤ Edit to display the Figures Editor.

2   Select the figure(s) to delete, right-click, and click Delete. The figure icon  is replaced with a  and the row is set to read-only and cannot be edited.

3   To undo the deletion for a single row or multiple rows, select the row(s), right-click, and click Undelete. To undo all the deletions in the vista, click  .

4   To apply the deletion for a single row or multiple rows, select the row(s), right-click, and click Apply. To apply all deletions and changes in the vista, click  .

**To delete a figure in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the Command line, enter:
DELETE FIGURE <name>.

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Figures collection ➤ right-click <named> Figure ➤ Delete

Survey Command Window Syntax

DELETE FIGURE (name)

Command Synonyms

DEL FIG

| Parameter | Definition |
|-----------|-----------|
| name | The name of the figure that you want to delete. |

# Editing Figure Elevations and Figure Geometry

Use the feature line editing commands to edit survey figure elevations and figure geometry.

You can access these commands by clicking Modify tab ➤ Edit Geometry Panel or Modify tab ➤ Edit Elevations panel. For more information on these commands see Editing Feature Lines (page 751).

# Figure Inquiry Commands

Use either the Figure collection in the Survey Toolspace or the Survey Command window to access Figure Inquiry commands to identify figures, obtain closure information, and determine the area and perimeter for the figures in your drawing.

# Identifying a Figure

Identify and display the figure name for a selected figure.

**To identify a figure**

**1** In Toolspace, on the Survey tab, expand the Figures collection.

**2** Right-click a figure ➤ Show Properties. The properties for the figure are displayed in the list view.

**To identify a figure in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the Command line, enter:
ID FIG

The following prompt is displayed:

Select figure:

After you select the figure, the figure name displays as shown below:

Name: {Figure Name}

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

ID FIG

Command Synonyms

DISP FIGS, LIST FIGS, DSP FIGS

# Displaying All Figures

Use either the Survey Toolspace or the Survey Command Window Display All command to view a list of all the existing figure names in a current drawing.

**To display all the figures in the current drawing in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the Command line, enter:
    DISPLAY FIGS

**Command Line Example**

The following is an example of a typical list generated by the Display Figures command:

Figure:

PL

Alder

Drive

Drive 2

Wall

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

DISPLAY FIGURES

Command Synonyms

DISP FIGS, LIST FIGS, DSP FIGS

## Listing Mapcheck Information for a Figure

Use the Mapcheck command to check your figure for length, course, perimeter, area, error of closure, or precision information.

Calculate closure, precision, perimeter, and area using the listed distances, courses, and units based on the current drawing unit precision set.

**NOTE** Use the Inverse command to return exact distances and courses, based on the current survey database distance units in the Survey Database Settings.

### Notes on Mapcheck Information

The Mapcheck report starts at the beginning of the figure. For each segment, based on the Linear and Angle precision (set in the Survey Database Settings) then the next figure vertex XY coordinates are computed based on the inversed direction and distance/curve data (rounded to the Linear and Angle precision values). For closed figures, due to the fact that error is introduced into the sequential computation of vertices of the mapcheck report, a closure error, closure direction, and precision can be calculated. The area is also based on the computed vertex XY coordinates.

### To do a Mapcheck in Toolspace

■ In Toolspace, on the Survey tab, expand the Figures collection, right-click a figure, and click Display Mapcheck. The Figure Display vista is displayed with the Mapcheck information for the selected figure.

### To Mapcheck a figure in the Survey Command Window

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:

MAPCHECK [figure name]

For more information about performing mapcheck on labels, see Performing a Mapcheck Analysis (page 1608).

### Command Line Example

The following is a typical report generated by the Mapcheck command:

```
Line Length: 73.01          Course: N 00-00-00 E

Line Length: 92.99          Course: N 90-00-00 W

Line Length: 73.09          Course: S 02-36-16 E

Line Length: 89.67          Course: S 90-00-00 E

Perimeter: 328.76

Area: 6667.83                       0.15 acres

Mapcheck Closure - (Uses listed courses & COGO Units)

Error Closure: 0.005          Course: 285-27-17

Precision 1: 70392.06
```

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

MAPCHECK FIGURE [figure]

Command Synonyms

MAPCHK FIG
CHECK FIG
CHK FIG

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure you want to mapcheck. |

# Listing Inverse Information for a Figure

Use the Inverse command to display the inverse information which includes the length, course, perimeter, area, closure error, and precision information for a given figure.

Closure, perimeter, precision, and area values are calculated using the exact XY coordinate data of each figure vertex.

### Notes on Inverse Information

The Inverse report starts at the beginning of the figure. For each segment, the inverse report lists the direction and distance, or curve data computed from the XY coordinates of the endpoints of the figure segments. The area is calculated from the XY coordinates of each segment.

### To display Inverse (exact closure) information using Toolspace

■ In Toolspace, on the Survey tab, expand the Figures collection. Right-click a figure ➤ Display Inverse. The Figure Display vista is displayed with the Inverse information for the selected figure.

> **NOTE** Figures that have vertices created using Point names will display the point name in the Point column.

### To calculate the mathematical closure of a figure using the Survey Command Window

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
INVERSE FIGURE [figure]

### Command Line Example

The following is a typical report generated by the Mapcheck command:

```
Figure Name: WALL

North: 5000.000            East: 5000.000

Line Length: 431.47           Course: 355-00-19

North: 5429.830           East: 4962.435
```

```
Line Length: 265.59          Course: 43-52-25

Line Length: 609.71          Course: 93-31-48

North: 5583.743        East: 5755.061

Line Length: 954.40          Course: 232-17-32

North: 5000.000        East: 5000.000

Perimeter: 2261.16

Area: 234948.16                    5.39 acres

Mathematical Closure - (Uses Survey Units)

Error Closure: 0.000          Course: 0-00-00

Error Closure: 0.000          Course: 0-00-00

Precision 1: 2261164601.60
```

## Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

INVERSE FIGURE [figure]

Command Synonyms

INV FIG

| Parameter | Definition |
|-----------|------------|
| figure | The name of the figure you want to inverse. |

# Listing the Area and Perimeter of a Figure

Determine and report the area and perimeter of an existing figure. Use the Toolspace list view to list the figure properties, or you can use the Area Perimeter command. The figure can be any shape, provided it is a closed figure.

**NOTE** The area and perimeter information is also listed at the top of the Figure Display vista when displaying Inverse information for the figure.

**To list the area of a figure**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the Command line, enter:
AREA [figure]

**Command Line Example**

The following is a sample valid entry using the Area command:

BEG TRIANGLENE 1 0 0NE 2 10 10NE 3 0 10CloseAREA TRIANGLEArea = 50.000 Perimeter = 46.5028

### Quick Reference

Toolspace

Survey tab: Survey Databases ➤ <named survey database> ➤ Network ➤ right-click <named> Network ➤ Survey Command Window

Survey Command Window Syntax

AREA [figure]

| Parameter | Definition |
| --- | --- |
| figure | The name of the figure for which you want to list the area and the perimeter. |

# Labeling Figures

**To label survey figures**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Figure ➤ Add Figure Labels 🏷 .

2  In the Add Labels dialog box, specify the label type and label style.

**To add labels to single or multiple segments**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Figure ➤ Add Single Segment or Multiple Segments 🏷 .

2  If you select Single Segment you are prompted to select a point on an entity in the drawing. If you select Multiple Segment you are prompted to select an entity in the drawing.

**To add tables**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Figure ➤ Add Line/Curve/Segment 🔲 .

2  Specify values in the Table Creation dialog box.

For more information, see Add Labels Dialog Box (page 1970) and Table Creation Dialog Box (page 2441).

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Figure ➤ Add Figure Labels 🏷

Menu

Survey menu ➤ Add Labels
Survey menu ➤ Add Tables

# Survey Analysis and Output

# 13

You can use analysis tools, such as Least Squares, to make adjustments in Survey Networks as well as individual traverses. Use the menus on the Survey Command Window to input survey data using the command line.

## Least Squares Analysis

You can use Least Squares Analysis to make adjustments in Survey Networks.

A surveyor's fundamental job is making measurements, or observations, to determine the relative position of points. In doing this, many surveyors typically make more measurements than are minimally necessary to rule out the possibility of gross errors, or blunders, which might have occurred during the measurement process. As a result, redundant measurements, together with random measurement errors, create a model that is geometrically over-determined. This condition may also be referred to as an over-constrained problem.

You can calculate the most probable value for each observation by adjusting each of the observations simultaneously so that the sum of the squares of the residuals (the difference between measured and adjusted observations) is at a minimum. This is the origin of the term "least squares."

You use the AutoCAD Civil Survey Least Squares commands to perform least squares adjustments of surveys. Traverse loops and networks of distance and angle measurements are evaluated by the observation equation method.

## Using Least Squares with Networks

Use the Least Squares method to make adjustments to the survey observations database. These adjustments update points in the survey database and the line work input with your survey.

### Creating an Input File for a Network

The Create Input File command creates the Least Squares Network input file, *network.lsi*, which is generated from station-to-station observations in the current project survey database (sideshots are not considered).

**To create an input file for a network**

■ In Toolspace, on the Survey tab, expand the Networks collections. Right-click a named network ➤ Least Squares Analysis ➤ Create Input File.

---

**NOTE** The current equipment settings are used to compute the standard deviations for each observation, when the observation is created.

---

## Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ right-click <named> network ➤ Least Squares Analysis ➤ Create Input File

## Editing the Input File for a Network

Use the Edit Input File command to add or make changes to the data using the text editor you specified in the External Editor Settings.

**To edit an input file**

1 In Toolspace, on the Survey tab, expand the Networks collection. Right-click a named network ➤ Least Squares Analysis ➤ Edit Input File.

2 In the text editor, enter your changes.

3 Click File menu ➤ Save. Click Exit to leave the text editor and return to the drawing.

## Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ right-click <named> network ➤ Least Squares Analysis ➤ Edit Input File

## Processing the Input File for a Network

Use the Process Input File command to process and view the results of the Least Squares adjustment for the network.

The program reads the input file, *network.lsi*. The adjustment is calculated, and the program creates a least squares network adjustment file (*network.adj*) and a least squares network output file (*network.lso*).

**To process the input file for a network**

1 In Toolspace, on the Survey tab, expand the Networks collection. Right-click a named network ➤ Least Squares Analysis ➤ Process Input File.

2 Use the scroll bar to review the information in the file. You can also save the output file to a new filename, or print the file to your printer.

The least squares network output file initially displays the Survey Least Squares calculations data, based on the processed *network.lsi* input file.

3 Page down to display the observation data for the network.

The measured observations and standard deviations are the values that existed before the Least Squares adjustment. The adjusted observations and residuals are the results of the Least Squares adjustment.

4 Page down again to display the adjusted coordinates for the network. The point identifiers, northing, easting, northing standard deviation, and easting standard deviation for each unknown point are listed.

5 Page down to the end of the file to display the error analysis for the network. The values of the semi-major and semi-minor axes, as well as the axis azimuth of the error ellipse for each unknown point are listed.

## Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ right-click <named> network ➤ Least Squares Analysis ➤ Process Input File

# Displaying the Output File for a Network

Use the Display Output File command to view information about a network. The least squares network output file, *network.lso*, is automatically loaded into the text editor.

**To display the output file for a network**

1 In Toolspace, on the Survey tab, expand the Networks collections. Right-click a <named> network ➤ Least Squares Analysis ➤ Display Output File.

2 Use the scroll bar to review the information in the file. Optionally, from the viewer, save the output file to a new file name, or print the file to your printer.

## Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ right-click <named> network ➤ Least Squares Analysis ➤ Display Output File

# Updating the Survey Database with Adjusted Coordinates

Use the Update Survey Database command to import the adjusted network points in the survey database and their graphic information into the current drawing. The Least Squares network adjustment file, *network.adj*, is automatically selected.

**To update the database with the adjusted network coordinates**

1 In Toolspace, on the Survey tab, expand the Networks collections. Right-click a <named> network ➤ Least Squares Analysis ➤ Update Survey Database.

2 In the Least Squares Adjusted File Selection dialog box, select the *network.adj* file.

When you update the survey database and insert the network into the drawing, the network object can display error ellipses according to the Style properties for the network.

The following illustration shows the least squares network:
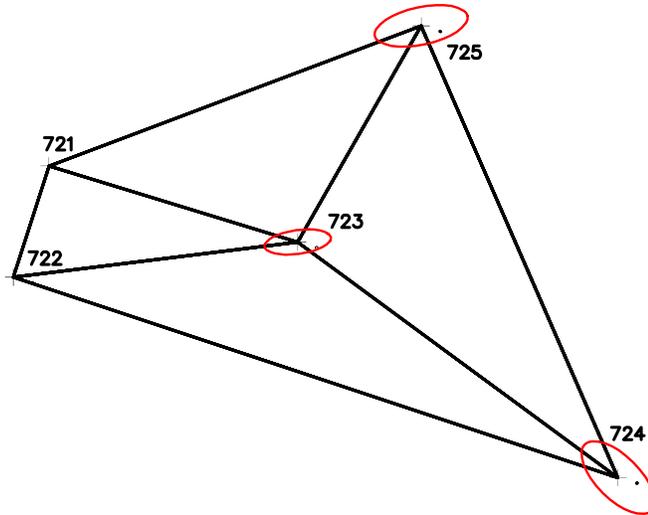
### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ right-click <named> network ➤ Least Squares Analysis ➤ Update Survey Database

## Least Squares Formulas

You can use these formulas as a reference for Least Squares analysis.

## Least Squares Algorithm

The algorithm used by the AutoCAD Civil Survey Least Squares program uses a series of matrixes and formulas.

This algorithm is used to determine the probable position of points based on the least squares method.

The following matrix equation, based on a system of weighted observations, is solved iteratively:

$$X = (A^T P A)^{-1} A^T P L$$

where:

- X is a matrix that contains the difference between the current coordinates and the new coordinates of each unknown station, and that updates the current coordinates with each iteration (until X @ 0.0).

- A is a matrix that is created from the observations and coordinates of the points, based on a linearization by a Taylor series expansion of the specific observation type.

- P is a diagonal matrix with the standard deviations, or weights, of the observations.

- L is a matrix that contains the difference between measured observations and calculated observations.

# Horizontal Survey Adjustment

The first step in the adjustment of a horizontal survey is the calculation of C, the coordinate matrix, with the approximate coordinates for all stations.

$$C = \begin{array}{c} \\ \\ \\ \\ \\ \\ \\ m \end{array} \begin{vmatrix} x_1 & y_1 \\ x_2 & y_2 \\ x_3 & y_3 \\ . & . \\ . & . \\ . & . \\ x_m & y_m \end{vmatrix}^1$$

Next, the L matrix is calculated based on the calculated observations derived from the coordinate matrix, C, and the measured observations. The following formula is used to determine each Li observation value:

$$L_i = O_{meas} - O_{calc}$$

where:

- Omeas = the observation as measured

- Ocalc = the observation as calculated

The L matrix appears as follows:

$$L = \begin{array}{c} \\ \\ \\ \\ \\ \\ \\ m \end{array} \begin{vmatrix} L_1 \\ L_2 \\ L_3 \\ . \\ . \\ . \\ L_m \end{vmatrix}^1$$

Next, the diagonal weight matrix, P, is calculated based on the standard deviations of distances and angles.

The following formulas are used to solve for $P_{L_{ij}}$ , the weighted distance value, and $P_{\theta_{ijk}}$ , the weighted angular value:

$$P_{L_{ij}} = \frac{1}{(S_{L_{ij}})^2}$$

where:

$S_{L_{ij}}$ is the standard deviation of the length ij, and

$$P_{\theta_{jik}} = \frac{1}{(S_{\theta_{jik}})^2}$$

where:

$S_{\theta_{jik}}$ is the standard deviation of angle jik

$$P = \begin{array}{c}{}\\{}\\{}\\{}\\{}\\{}\\{}\\{}\\{m}\end{array}\left|\begin{array}{cccccc} p_1 & & & & 0 & 0 \\ & p_2 & & & & 0 \\ & & p_3 & & & \\ & & & . & & \\ & & & & . & \\ 0 & & & & . & \\ 0 & 0 & & & & p_m \end{array}\right|^{1}$$

The final step is to calculate the A matrix, which relates the changes in coordinates to each observation determined through the use of distance observation equations and angle or azimuth/bearing observation equations. These equations are initially nonlinear, so the Taylor series approximation is used to render them as linear equations.

## Distance Observation Equation Geometry

The change in coordinates for each observation is expressed in the following Taylor series approximations of the distance observation equation:

$$dx_i = \frac{X_i - X_j}{L_{ij}}$$

$$dy_i = \frac{Y_i - Y_j}{L_{ij}}$$

and

$$dy_i = \frac{Y_i - Y_j}{L_{ij}}$$

$$dy_j = \frac{Y_j - Y_i}{L_{ij}}$$

where

■ Lij = the observed length of line IJ

■ Xi, Yi = the most probable coordinates of point I

■ Xj, Yj = the most probable coordinates of point J

The following illustration shows the geometry of the distance observation equation:
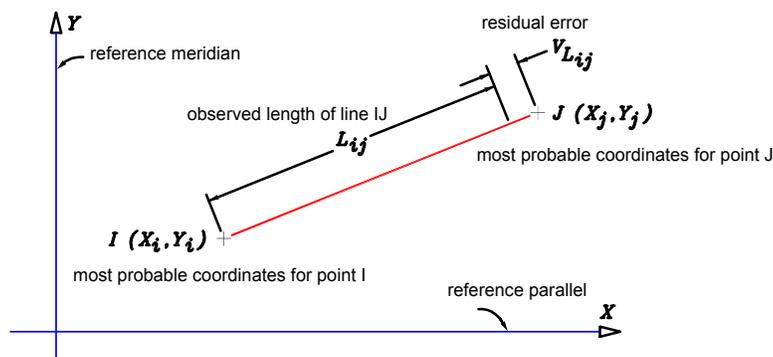


## Angle Observation Equation Geometry

The change in coordinates for each observation is expressed in the following Taylor series approximations of the angle observation equation:

$$dx_i = -dx_j - dx_k$$

$$dy_i = -dy_j - dy_k$$

$$dx_j = \frac{Y_i - Y_j}{(L_{ij})^2}$$

$$dy_j = \frac{X_i - X_j}{(L_{ij})^2}$$

$$dx_k = \frac{Y_i - Y_k}{(L_{ik})^2}$$

$$dy_k = \frac{X_i - X_k}{(L_{ik})^2}$$

where

- Lij = the observed length of line IJ

- Lik = the observed length of line IK

- Lik = the observed length of line IK

- Xi, Yi = the most probable coordinates of point I

- Xj, Yj = the most probable coordinates of point J

- Xk, Yk = the most probable coordinates of point K

The following illustration shows the geometry of the angle observation equation:
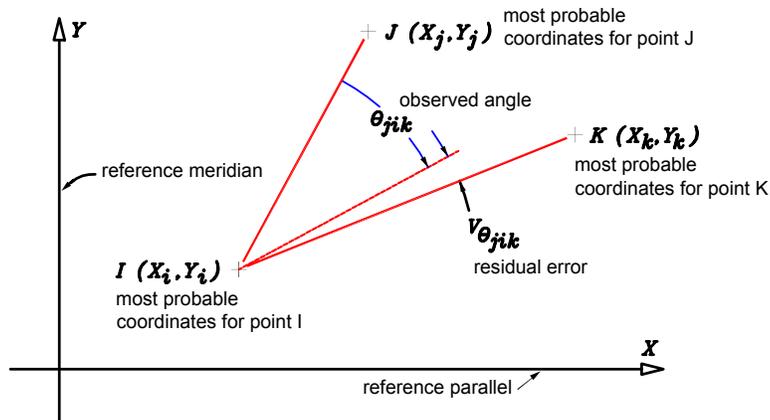
## Azimuth/Bearing Observation Equation Geometry

The change in coordinates for each observation is expressed in the following Taylor series approximations of the azimuth/bearing observation equation:

$$dx_i = \frac{X_i - X_j}{(L_{ij})^2}$$

$$dy_i = \frac{Y_i - Y_j}{(L_{ij})^2}$$

and

$$dx_j = \frac{X_j - X_i}{(L_{ij})^2}$$

$$dy_j = \frac{Y_j - Y_i}{(L_{ij})^2}$$

where

- Lij = the observed length of line IJ

- Xi, Yi = the most probable coordinates of point I

- Xj, Yj = the most probable coordinates of point J

The following illustration shows the geometry of the azimuth/bearing observation equation:
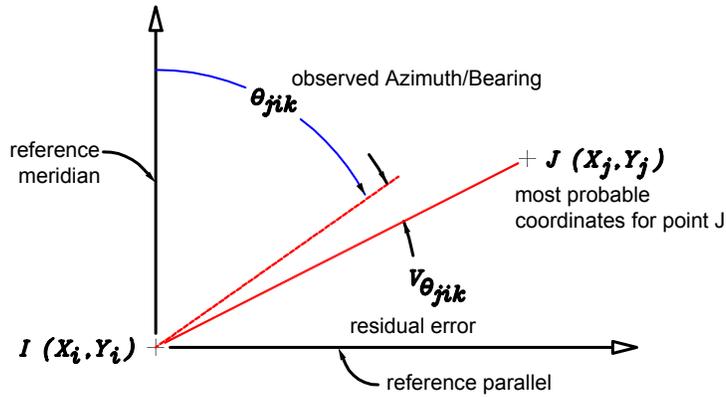
## Weighted Observations

The following calculated values are used in the A matrix:

$$A = \begin{vmatrix} dx_1 \ dx_2 \ dy_1 \ dy_2 \ ... \\ dx_1 \ dx_2 \ dy_1 \ dy_2 \ ... \\ . \\ . \\ . \end{vmatrix}^{2}_{m}$$

The computed matrices A, P, and L are now plugged into the matrix formula for a system of weighted observations to solve for the X matrix, as stated previously:

$$X = (A^{T}PA)^{-1}A^{T}PL$$

This results in the X matrix, that consists of the difference between the current coordinates and the most probable values for the coordinates:

$$X = \begin{vmatrix} \Delta x_1 \\ \Delta x_2 \\ \Delta y_1 \\ \Delta y_2 \\ . \\ . \\ . \\ \Delta x_n \\ \Delta y_n \end{vmatrix}^{1}_{2n}$$

These values are used to revise the coordinate matrix, C. The iteration process continues, recalculating the A, P, and L matrices and solving for the X matrix until the values in the X matrix are negligible.

After the X matrix values have reached the desired limits, the V matrix is calculated. This matrix consists of the residuals after adjustment and is solved for by the following matrix equation:

$$V = AX - L$$

This results in the V matrix, which appears as follows:

$$V = \begin{array}{c} \\ \\ \\ \\ \\ \\ \\ \\ m \end{array} \begin{vmatrix} v_1 \\ v_2 \\ v_3 \\ . \\ . \\ . \\ v_m \end{vmatrix}^{1}$$

## Chi-Square Value and Goodness-of-Fit Test

Chi-square distribution is a probability tool used in hypothesis testing. In the case of least squares adjustment, original observations are compared to adjusted observations and evaluated in terms of how widely these observations vary with respect to each other.

The following formula is used to calculate X2, the chi-square value:

$$\chi^2 = \sum_{i=1}^{m} v_i^2 p_i^2$$

where

■  m = the number of observation equations

■  vi = the residual value from the V matrix

■  pi = the corresponding weight from the P matrix

The X2 value is small if the adjusted observations are close to the corresponding measured observations. This is referred to as a "good fit." A goodness-of-fit test is then conducted in which the X2 value is compared to those from a table of critical values of chi-square distribution at the 5% level. A X2 value below that listed in the table passes, while a larger X2 value than that listed fails, indicating possible blunders in the initial surveying measurements.

## Standard Deviation Computations

Calculation of the covariance matrix, Q, is next. The covariance matrix consists of the coefficients of the unknowns from the observation equations, and is used to compute standard deviations and error ellipses. The following matrix formula, using the previously solved A and P matrices, is used:

$$Q = (A^TPA)^{-1}$$

The Q matrix appears as follows:

$$Q = \begin{vmatrix} n \\ S_1^2 & S_1 S_2 & \cdots & S_1 S_n \\ S_2 S_1 & S_2^2 & \cdots & S_2 S_n \\ & & \cdot & \\ & & \cdot & \\ & & \cdot & \\ {}_n S_n S_1 & S_n S_2 & \cdots & S_n^2 \end{vmatrix}^n$$

Next, the calculation of the degrees of freedom in an adjustment, r, is accomplished through the use of the following formula:

$$r = m - n$$

where

- m = the number of observation equations

- n = the number of unknowns (or 2n in the case of coordinates, since both x and y are unknown)

The next procedure is the calculation of the standard deviation of unit weight for a weighted adjustment, So, which is done through the following matrix formula:

$$S_o = \sqrt{\frac{(V^T P V)}{r}}$$

where

- r = the degree of freedom in the adjustment

Calculation of the standard deviations of the individual adjusted quantities, $S_{x_i}$, is next. These are determined by the following formula:

$$S_{x_i} = S_o \sqrt{Q_{x_i x_i}}$$

where

- So = the standard deviation of unit weight

- $S_{x_i}$ = the diagonal element in the ith row and ith column from the covariance matrix Q

## Error Rectangles and Ellipses

After the completing the Least Squares adjustment, the covariance matrix is used to calculate the positional standard deviations $S_{x_i}$ and $S_{y_i}$.

The following illustration shows that standard deviations represent half-dimensions of the standard 68% probability error rectangle around each point:
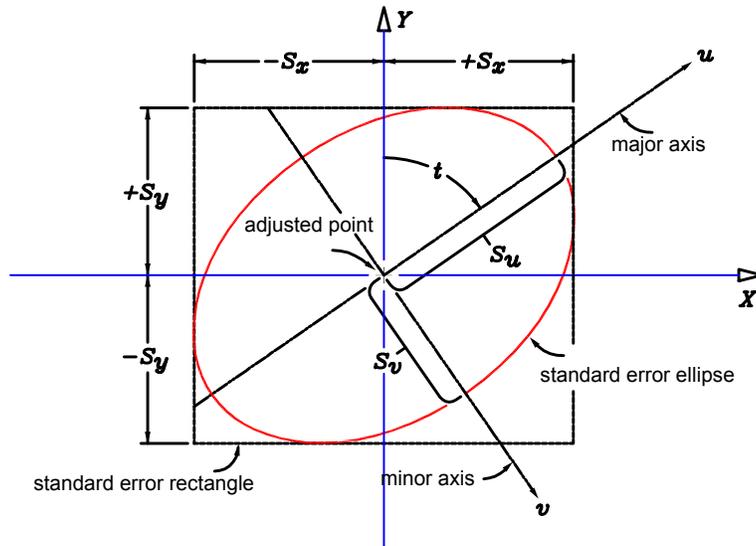
Where:

- t = the orientation of the error ellipse, the direction of the semi-major axis

- u = semi-major axis of the error ellipse

- v = the semi-minor axis error ellipse

- x = half-width dimension of the error rectangle

- y = half-height dimension of the error rectangle

- S = Positional standard deviation of a point

## Standard Error Rectangle and Ellipse Geometry

To further refine this process, a standard error ellipse can also be drawn. Using an F distribution from applied statistics, either a 95% or 99% probability error ellipse is created. This ellipse has arcs tangent to the sides of the error rectangle, with its orientation determined by an angle, t, and auxiliary orthogonal u-v axes. Again, values from the covariance matrix are used to calculate the t angle and the associated semi-major and semi-minor axes.

First solve three variables to use in equations that determine the semi-major and semi-minor axes: K, Quu and Qvv.

The following formulas are used:

$$K = \sqrt{[(Q_{yy} - Q_{xx})^2 + 4(Q_{xy})^2]}$$

$$Q_{uu} = \frac{(Q_{yy} + Q_{xx} + K)}{2}$$

and

$$Q_{vv} = \frac{(Q_{yy} + Q_{xx} - K)}{2}$$

where

■ Qxx, Qyy, and Qxy are values from the covariance matrix.

Then, these values are used to solve two equations to determine Su, the semi-major axis length and Sv, the semi-minor axis length:

$$S_u = S_o \sqrt{Q_{uu}}$$

and

$$S_v = S_o \sqrt{Q_{vv}}$$

where

■ So = standard deviation of unit weight

Finally, the angle t, which the u axis makes with the Y axis, is solved for by the following equation:

$$\tan 2t = \frac{2Q_{xy}}{Q_{yy} - Q_{xx}}$$

The adjusted point is most likely found within this ellipse with either a 95% or 99% degree of confidence.

# Traverses

Use the Traverses collection to manage the existing traverses defined within the network.

You can create a traverse from data in a field book file, or use the Traverse Editor to manually enter survey data, for example hand notes or data from a site plan. Using the Traverse Editor is an efficient way to view and edit all the setups in a traverse contained in a Network.

## Creating Traverses

Create traverses to perform traverse closure analysis such as Compass Rule, Crandall Rule, or Transit Rule. You can also perform a least squares analysis on a traverse.

You can create a traverse from data that you import from a field book file, or use the Traverse Editor to manually enter traverse observations.

**To create a traverse**

1 In Toolspace, on the Survey tab, expand the Networks collections. Expand a network ➤ right-click Traverses ➤ New.

2 In the New Traverse dialog box, click a cell in the Value column and enter values. Click OK.

**NOTE** Observations created using Point names will display in the Value column.

### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ <named network> ➤ Traverses ➤ New

Dialog Box

Traverse Editor (page 2403)

## Editing Traverse Properties

Change the initial properties for a traverse in the Traverse Properties dialog box.

**To edit traverse properties**

1  In Toolspace, on the Survey tab, expand the Networks collections. Expand a named network ➤ Traverses.

2  Right-click a named traverse ➤ Properties.

3  In the Traverse Properties dialog box, click a cell in the Value column and make any necessary changes.

4  Click OK.

### Quick Reference

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ <named network> ➤ Traverses ➤ <named> traverse ➤ right-click Properties

Dialog Box

New Traverse (page 2419)

## Editing Traverses

Edit the properties of all traverses using the Traverses Editor panorama window, or edit an individual traverse in the Traverse Editor.

**To edit traverses**

1  In Toolspace, on the Survey tab, expand the Networks collections. Expand a named network ➤ Traverses ➤ Edit.

2  In the Traverses Editor Panorama window, click a cell and enter changes.

3  Right-click a cell and click New to create a traverse or click Delete to remove an existing traverse.

**To edit an individual traverse using the Traverse Editor**

1  In Toolspace, on the Survey tab, expand the Networks collections. Expand a named network ➤ Traverses ➤ right-click named traverse ➤ Edit.

2  In the Specify Initial Setup dialog box, enter the Initial Station and the Initial Backsight. Click OK.

**3** If you have defined the traverse, the current Setup property list is displayed on the left side of the panorama. In the current Setup property list you can edit the Backsight Orientation, Backsight Face1 and BacksightFace2 settings, and the Instrument Height.

> **NOTE** If the Setup you select is the initial setup for the traverse and the backsight point is not defined by a control point, you can enter the backsight direction. If the backsight direction is not assigned in the setup properties, you can enter it as a backsight observation.

**4** Expand each Station point. Enter or edit the observations for each Setup. A new blank observation row is added to the last setup observation where you can enter a new backsight or foresight observation.

> **NOTE** Observations created using Point names will display the name in the Name column.

**5** Specify a backsight observation in a new row by entering the point identifier for the setup.

**6** Specify a foresight observation to the next station point by entering a point identifier that has not been referenced in the current traverse.

When you specify the new foresight point identifier, the next Station point and setup are created.

> **NOTE** Once you have referenced a foresight point identifier, all new foresight observations must use the same point identifier.

**7** Click 💾 to apply the changes.

## Quick Reference

Toolspace

Survey tab: survey Survey Databases ➤ <named> survey database ➤ Networks ➤ <named network> ➤ Traverses ➤ right-click Edit
Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ <named network> ➤ Traverses ➤ <named> traverse ➤ right-click Edit

Dialog Box

Traverse Editor (page 2403)

# Performing Traverse Analysis

Analyze traverse data to determine the error of closure and to update the observations and survey data that is changed by the traverse adjustment.

**To perform traverse analysis**

**1** In Toolspace, on the Survey tab, expand the Networks collections. Expand a named network ➤ Traverses ➤ right-click named traverse ➤ Traverse Analysis.

**2** Review the settings in the Traverse Analysis dialog box. Enter any changes and click OK.

**Quick Reference**

Toolspace

Survey tab: expand Survey Databases ➤ <named> survey database ➤ Networks ➤ <named network> ➤ Traverses ➤ <named> traverse ➤ right-click Traverse Analysis

Dialog Box

Traverse Analysis Dialog Box (page 2408)

# Mapcheck Analysis

Perform a Mapcheck Analysis by selecting AutoCAD Civil line and curve labels to determine values from label objects based on the precision of the annotation of the label object, or enter mapcheck data manually.

For more information, see Mapcheck Analysis (page 1607).

# Displaying Point Information

You can use the Point Information commands to list angle, distance, and bearing information about the points that exist in the survey database.

# Listing the Horizontal Angle Between Points

Display the angle between any three points in the survey database. You must specify a backsight point, instrument point, and ahead point.

**To list the horizontal angle between three points in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  In the Survey Command Window, click Point Information menu ➤ Angle.

3  Enter the point identifier of the backsight.

4  Enter the point identifier of the instrument point.

5  Enter the point identifier of the foresight point.

**To list the horizontal angle between three points in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the command line, enter:
ANGLES [point 1] [point 2] [point 3]

**Command Line Example**

NE 1 1200 1200

NE 2 1000 1000

NE 3 800 1100

ANGLE 1 2 3

Angle: 108-26-06

### Quick Reference

Survey Command Window Menu

 Point Information ➤ Angle

Survey Command Window Syntax

 ANGLES [point 1] [point 2] [point 3]

Command Synonyms

 ANGLE
 ANGS
 ANG
 A

| Parameter | Definition |
|-----------|------------|
| point 1 | The point identifier of the backsight. |
| point 2 | The point identifier of the station. |
| point 3 | The point identifier of the foresight. |

## Listing the Bearing Between Two Points

Use the Bearing command to list the bearing from one point to another.

**To list a bearing between two points in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 In the Survey Command Window, click Point Information menu ➤ Bearing.

3 Enter the point identifier of the start point.

4 Enter the point identifier of the ahead point.

**To list a bearing between two points at the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the command line, enter:
 BEARINGS [point 1] [point 2] or
 B [point 1] [point 2]

**Command Line Example**

The following example shows a valid entry using the Bearing command:

NE 1 1000 1000

NE 2 2000 2000

B 1 2

Bearing: N 45-00-00 E

The bearing between point 1 and point 2 is listed.

The following example shows a valid entry using the Bearing command to define an observed bearing. Angles are in the current survey database units.

NE 1 1000 1000

NE 2 2000 2000

B 1 2 50 1

Observed direction: N 50-00-00 E

## Quick Reference

Survey Command Window Menu

  Point Information ➤ Bearing

Survey Command Window Syntax

  B [point 1] [point 2] [observed bearing] [quadrant]

Command Synonyms

  BEARINGS
  BRGS

| Parameter | Definition |
| --- | --- |
| point 1 | The point identifier of the origin of the bearing. |
| point 2 | The point identifier to which the bearing is measured. The bearing is measured from point 1 to point 2. |
| observed bearing | The observed bearing between point 1 and point 2. |
| quadrant | The quadrant for the bearing. Specify the quadrant using one of the following numbers: 1 (for NE), 2 (for SE), 3 (for SW), or 4 (for NW). |

# Listing the Azimuth Between Two Points

You can list the azimuth between two points in the survey database using the Azimuth command.

**To list the azimuth between two points in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 In the Survey Command Window, click Point Information menu ➤ Azimuth.

3 Enter the point identifier of the start point.

4 Enter the point identifier of the ahead point.

**To list the azimuth between two points at the Command line in the Survey Command Window**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 At the command line, enter:
AZIMUTH [point 1] [point 2] (observed azimuth)

**Command Line Example**

The following example shows a valid entry using the Azimuth command to list an azimuth. Angles are in the current survey database units.

NE 1 1000 1000

NE 2 2000 2000

AZ 1 2

Azimuth: 45-00-00

The following example shows a valid entry using the Azimuth command to define an observed azimuth.

NE 1 1000 1000

NE 2 2000 2000

AZ 1 2 50

Observed direction: 50-00-00

## Quick Reference

Survey Command Window Menu

   Point Information ➤ Azimuth

Survey Command Window Syntax

   AZIMUTH [point 1] [point 2] (observed azimuth)

Command Synonyms

   AZM
   AZ

| Parameter | Definition |
| --- | --- |
| point 1 | The point identifier of the origin of the azimuth. |
| point 2 | The point identifier to which the azimuth is measured. The azimuth is measured from point 1 to point 2. |
| observed azimuth | The observed azimuth between point 1 and point 2. |

# Listing the Distance Between Two Points

Use the Distance command to determine the horizontal distance between two points in the survey database.

**To determine the distance between two points in the Survey Command Window**

 1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

 2  In the Survey Command Window, click Point Information menu ➤ Distance.

 3  Enter the point identifier of the start point.

 4  Enter the point identifier of the ahead point.

**To determine the distance between two points at the Command line in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the command line, enter:
DISTANCE [point 1] [point 2]

**Command Line Example**

The following example shows a valid entry using the Distance command:

NE 1 1000 1000

NE 2 2000 2000

D 1 2

Distance: 1414.21

## Quick Reference

Survey Command Window Menu

Point Information ➤ Distance

Survey Command Window Syntax

DISTANCE [point 1] [point 2]

Command Synonyms

DIST
DIS
D
HDIST
HD
LENGTH

| Parameter | Definition |
|-----------|------------|
| point 1 and point 2 | The two existing points between which the distance is determined. |

# Listing the Direction and Distance Between Points

Use the Inverse Points command to determine the direction and distance between two points.

**To inverse points in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** In the Survey Command Window, click Point Information menu ➤ Inverse Points.

**3** Enter the point identifier of the start point.

**4** Enter the point identifier of the ahead point.

**To inverse points at the Command line in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the command line, enter:

INVERSE POINTS [point 1] [point 2]

**Command Line Example**

The information is based on the current display setting and is calculated from the lines between points one and two, two and three, and from three to one. The command then computes and displays the area.

The following example shows a valid entry using the Inverse Points command when units are set to feet:

NE 1 100 100

NE 2 100 200

NE 3 50 150

INV PTS 1 2

POINT 1 NORTH: 100.0000 EAST: 100.0000 ELEV: <Null>

Distance: 100.000 Course: N 90-00-00 E

POINT 2 NORTH: 100.0000 EAST: 200.0000 ELEV: <Null> 3

Distance: 70.710 Course: S 45-00-00 W Perimeter: 170.7

POINT 3 NORTH: 50.0000 EAST: 150.0000 ELEV: <Null> 1

Distance: 70.710 Course: N 45-00-00 W Perimeter: 241.4

POINT 1 NORTH: 100.0000 EAST: 100.0000 ELEV: <Null>

Area: 2500.00 0.06 acres

The following is an example of a valid metric entry using the Inverse Points command:

NE 1 100 100

NE 2 100 200

NE 3 50 150

INV PTS 1 2

POINT 1 NORTH: 100.0000 EAST: 100.0000 ELEV: <Null>

Distance: 100.000 Course: 100.0000

POINT 2 NORTH: 100.0000 EAST: 200.0000 ELEV: <Null> 3

Distance: 70.710 Course: 250.0000 Perimeter: 170.7

POINT 3 NORTH: 50.0000 EAST: 150.0000 ELEV: <Null> 1

Distance: 70.710 Course: 350.0000 Perimeter: 241.4

POINT 1 NORTH: 100.0000 EAST: 100.0000 ELEV: <Null>

Area: 2500.00 0.25 hectares

## Quick Reference

Survey Command Window Menu

Point Information ➤ Inverse Points

Survey Command Window Syntax

INVERSE POINTS [point 1] [point 2]

Command Synonyms

INV PNTS
INV PTS

| Parameter | Definition |
| --- | --- |
| point 1 and point 2 | The two existing points between which the relevant information displays. |

# Listing an Angle and Distance Using the Inverse Radial

You can use the Inverse Radial command to determine the angle and distance from a setup in a radial direction from the backsight point. You must specify a backsight point identifier, an instrument point identifier, and a foresight point.

**To list angle and distance information using the Inverse Radial in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** In the Survey Command Window, click Point Information menu ➤ Inverse Radial.

**3** Enter the point identifier of the backsight.

**4** Enter the point identifier of the instrument point.

**To list angle and distance information using the Inverse Radial at the Command line in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the command line, enter:
INVERSE RADIAL [backsight] [station]

**Command Line Example**

The following example shows a valid entry using the Inverse Radial command:

NE 1 100 100

NE 2 200 200

NE 3 175 300

NE 4 300 200

NE 5 310 220

NE 6 200 250

INVERSE RADIAL 2 1

Inverses reference instrument at point 2 backsighting point 1 3

Angle: 24-26-38 Distance: 213.600

POINT 3 NORTH: 175.000 EAST: 300.000 ELEV: <Null> 4 6

Angle: 341-33-54 Distance: 223.610

POINT 4 NORTH: 300.000 EAST: 200.000 ELEV: <Null>

Angle: 344-44-42 Distance: 241.870

POINT 5 NORTH: 310.000 EAST: 220.000 ELEV: <Null>

Angle: 11-18-36 Distance: 180.280

POINT 6 NORTH: 200.000 EAST: 250.000 ELEV: <Null>

The instrument is set on point 2, backsights point 1, and gives the angle turned from line 2-1 to line 2-3. This command returns the distance from point 2 to point 3, then does the same for points 4 through 6.

### Quick Reference

Survey Command Window Menu

Point Information ➤ Inverse Radial

Survey Command Window Syntax

INVERSE RADIAL [backsight] [station]

Command Synonyms

INV RADIUS
INV RAD

| Parameter | Definition |
|-----------|------------|
| backsight | The known reference point or line. |
| station | The point where the instrument is located (the occupied point). |

## Listing the Elevation Distance Between Two Points

Use the Vertical Distance command to determine the elevation difference between two points in the survey database.

**To list the elevation distance in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  In the Survey Command Window, click Point Information menu ➤ Vertical Distance.

3  Enter the point identifier of the start point.

4  Enter the point identifier of the ahead point.

**To list the elevation distance at the Command line in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the command line, enter:
VDIST [point 1] [point 2]

**Command Line Example**

The following example shows a valid entry using the Vertical Distance command:

NEZ 1 100 100 30.480

NEZ 2 200 200 33.528

VD 1 2

Vertical distance 3.048

This example determines the vertical distance from point 1 to point 2. A positive value indicates that point 2 is higher than point 1.

### Quick Reference

Survey Command Window Menu

Point Information ➤ Vertical Distance

Survey Command Window Syntax

VDIST [point 1] [point 2]

Command Synonyms

VD

| Parameter | Definition |
| --- | --- |
| point 1 and point 2 | These are the two existing points in the survey database. |

## Listing the Slope Distance Between Two Points

Use the Slope Distance command to determine the slope distance between two points.

**To determine the slope distance between two points in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   In the Survey Command Window, click Point Information menu ➤ Slope Distance.

3   Enter the point identifier of the start point.

4   Enter the point identifier of the ahead point.

**To determine the slope distance between two points at the Command line in the Survey Command Window**

1   In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2   At the command line, enter:
SDIST [point 1] [point 2]

The following illustration shows the Slope Distance command:
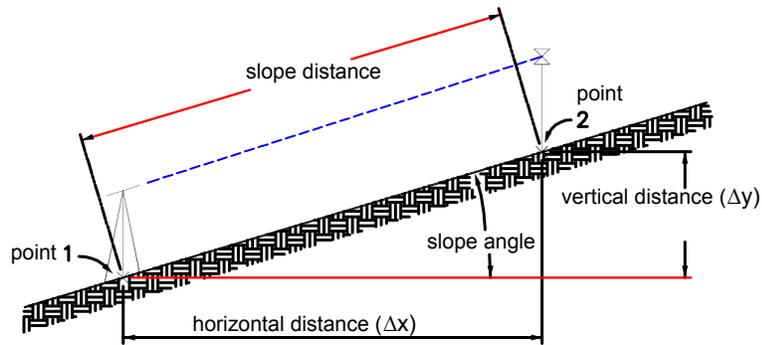
**Command Line Example**

The following example shows a valid entry using the Slope Distance command:

NEZ 1 100 100 30.480

NEZ 2 200 200 33.528

SD 1 2

Slope distance: 141.45

This example determines the slope distance from point 1 to point 2. Distances are in the current survey database units.

## Quick Reference

Survey Command Window Menu

  Point Information ➤ Slope Distance

Survey Command Window Syntax

  SDIST [point 1] [point 2]

Command Synonyms

  SD

| Parameter | Definition |
|---|---|
| point 1 and point 2 | These are the two existing points between which an invisible line is established. This line is used to determine the slope distance between the two points. |

# Listing the Grade Between Points

Use the Grade command to determine the grade between two points. You must specify a starting point and an ahead point.

**To list the grade between two points in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** In the Survey Command Window, click Point Information menu ➤ Grade.

**3** Enter the point identifier of the start point.

**4** Enter the point identifier of the ahead point.

**To list the grade between two points at the Command line in the Survey Command Window**

**1** In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

**2** At the command line, enter:

GRADE [point 1] [point 2]

The following illustrates the Grade command:
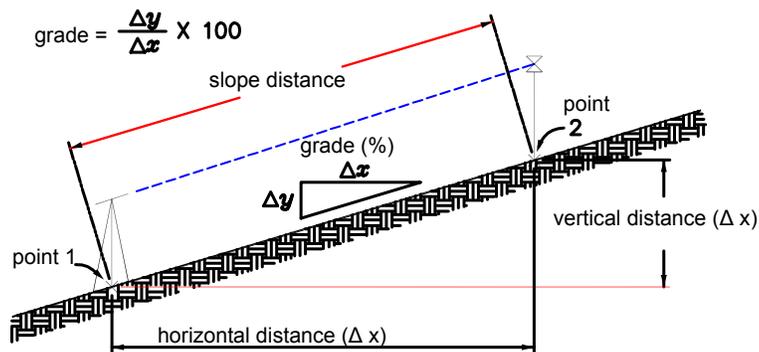


**Command Line Example**

The following example shows a valid entry using the Grade command:

NEZ 1 100 100 30.480

NEZ 2 200 200 32.004

GRADE 1 2

Grade: 1.0776 Percent

This example displays the grade from point 1 to point 2 in percent. When used as a built-in function, this command supplies the vertical angle in the current vertical reference system (zenith, nadir, or horizontal).

## Quick Reference

Survey Command Window Menu

Point Information ➤ Grade

Survey Command Window Syntax

GRADE [point 1] [point 2]

Command Synonyms

G

| Parameter | Definition |
|---|---|
| point 1 and point 2 | The two existing points between which the grade is determined. |

# Listing the Slope Angle Between Existing Points

Use the Slope Angle command to determine the slope angle between two points.

**To list the slope angle between two points in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  In the Survey Command Window, click Point Information menu ➤ Slope Angle.

3  Enter the point identifier of the start point.

4  Enter the point identifier of the ahead point.

**To list the slope angle between two points at the Command line in the Survey Command Window**

1  In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2  At the command line, enter:
   SLOPE [point 1] [point 2]

**Command Line Example**

The following example shows a valid entry using the Slope command:

NEZ 1 100 100 30.480

NEZ 2 200 200 33.528

SLOPE 1 2

Slope angle: 1.3719

This example determines the slope angle between point 1 and point 2. When used as a built-in function, this command supplies the vertical angle in the current vertical reference system (zenith, nadir, or horizontal).

## Quick Reference

Survey Command Window Menu

Point Information ➤ Slope Angle

Survey Command Window Syntax

SLOPE [point 1] [point 2]

| Parameter | Definition |
|---|---|
| point 1 and point 2 | The two existing points between which an invisible line is established. This line is used to determine the slope between the two points. |

# Outputting Survey Files

The Output file stores all the commands you enter at the Survey Command Line, including results from the query commands. The format of the output file is the same language or format as the Survey Command Language.

You can create, edit, or delete the output file using the commands in the Output menu. You can determine the name of the current output file using the Use Output File setting in the Survey User Settings dialog box.

The output file for each network in a survey database is saved to *Civil Projects\<named> Survey Database\<named> Network\Output.txt*.

**NOTE** For other options on outputting survey data see Exporting a Field Book File (page 243) and Exporting Survey Data to LandXML (page 244).

## Turning the Output File On or Off

Use the Output File menu to specify the settings that affect how the output file is used and updated during the Survey Command Window session.

**To turn the output file on or off**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 In the Survey Command Window, click Output File menu ➤ Use Output File. A check mark next to the item indicates it is active. Click the menu item to clear the check mark. Specify the following settings:

- Use Output File: a check mark next to the menu item indicates the Output File is active.

- Set File Name: enter a name for the output file.

- Use Point Course Echo: reports the course taken to reach a point, including the direction and the distance.

- Use Figure Course Echo: reports the course taken to create a figure, including direction and distance.

- Use Point Coordinate Echo: displays the point coordinates, including northing, easting, and elevation information.

- Use Figure Coordinate Echo: displays figure point coordinates, including northing, easting, and elevation information.

### Quick Reference

Survey Command Window Menu

Output File

## Viewing the Contents of the Survey Output File

Use the Output File menu to specify the settings that affect how the output file is used and updated during the Survey Command Window session.

**To view the output file**

1 In Toolspace, on the Survey tab, right-click a named network ➤ Survey Command Window.

2 In the Survey Command Window, click Output File menu ➤ View.

The Output.txt file is displayed in the text editor.

**Quick Reference**

Survey Command Window Menu

Output File ➤ View

# Astronomic Direction Calculations

You can use the Astronomic Direction Calculator to calculate astronomic directions from Sun Shot and Star Shot observations.

In the Astronomic Direction Calculator you can specify the calculation type as either Sun Shot or Star Shot. A Sun Shot calculates the astronomic direction from solar observations by the hour angle method and uses a multiple foresight procedure. A Star Shot calculates the astronomic direction from Polaris or star observations by the hour angle method and uses a single foresight procedure.

Each method requires a source of UT1 time (radio) and a stopwatch to measure the observations accurately. Set the stopwatch to zero (0) at UT1 time. Start the stopwatch at a known interval of UT1 time and write down the value of UT1 time (adding any necessary DUT corrections). When measuring the horizontal angle to the sun or star, record the stopwatch reading (stop time) in the field book along with the angular data.

### Single Foresight

This procedure is recommended for Polaris observations, or any sightings that have a large vertical angle inherent in the observation. The single foresight procedure uses the following sequence: direct on the target, direct on Polaris, reverse on target, reverse on Polaris. A stopwatch time is recorded for each pointing on the Polaris. The two stopwatch times and the four horizontal circle readings correspond to one set of data. It is recommended that you perform a minimum of three sets.

### Multiple Foresight

This procedure is used as a solar observation method. It uses the following sequence: direct on the target, multiple direct on the sun, multiple reverse on the sun (same number of times as direct), reverse on the target. A stopwatch time is recorded for each pointing on the sun. Pair up each observation on the sun (and its stop time) with its related target observation, and enter it into the editor as a single direct or reverse pair. The editor computes an azimuth (or bearing) for each pair. You can average any number of these azimuths. Do not average the multiple observations themselves because they are time-dependent and not a linear function of time.

# Calculating Directions with Sun Shots and Star Shots

In the Astronomic Direction Calculator you can select Sun Shot or Star shot to calculate astronomic directions.

Sun shots calculate a direction using solar observations by the hour angle method and use a multiple foresight method. Star Shots calculate a direction from Polaris or a star observation by the hour angle method and use a single foresight method.

**To calculate an astronomic direction using the Sun Shot or Star Shot calculations**

1 Click Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Astronomic Direction ☀ .

2 In the Calculation Type list, select either Sun Shot Calculation, or Star Shot Calculation.

**3** Under Observation Station Data, specify the following:

- Station Point

- Backsight Point

- Station Latitude

- Station Longitude

- UT1 Time (Sun Shots only)

See Astronomic Direction Calculator Dialog Box (page 2424) for more information.

**4** Under Ephemeris Data, enter the following:

- GHA 00 Hours

- GHA 24 Hours

- Declination 00 Hours

- Declination 24 Hours

- Sun Semi-diameter (Sun Shots only)

---

**NOTE** Enter zero (0) if you are sighting either the center or both the trailing and leading edges of the sun. The left edge is always the leading edge at latitudes greater than 23.5 degrees north and greater than 23.5 degrees south. If only the left edge is being sighted (left when facing the sun), then the semi- diameter is positive; likewise, the semi-diameter is negative when only the right edge is sighted.

---

**5** Click ⬛ to create an observation set. Enter Direct and Reverse observations for the Backsight Observation, Sun Observationor Star Observation and the Stop Time.

The observations determine the true astronomic direction from the first setup to the backsight. When you enter the stop time for an observation, the observed and average directions are calculated automatically.

## Quick Reference

Ribbon

Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Astronomic Direction ⬛

Menu

Click Surveymenu ➤ Astronomic Direction Calculator.

Command Line

SurveyCalculator

Dialog Box

Astronomic Direction Calculator (page 2424)

# Survey Command Reference

# 14

You can use commands to quickly access survey functionality.

## Survey Commands

The following table lists the survey-related AutoCAD Civil commands that can be entered at the command line.

| Command | Description |
| --- | --- |
| CST | Closes the Toolspace Survey tab (page 98) |
| BrowseToSvFigure | Synchronizes the Toolspace Survey tab to the specified figure in the drawing (page 349) |
| BrowseToSvNetwork | Synchronizes the Toolspace Survey tab to the specified network in the drawing (page 203) |
| BrowseToSvObservation | Synchronizes the Toolspace Survey tab to the specified observation in the drawing (page 263) |
| CreateFigureFromObject | Creates a survey figure from AutoCAD and AutoCAD Civil objects (page 323) |
| EditSvDatabaseSettings | Displays the Survey Database Settings dialog box (page 216) |
| EditSvUserSettings | Displays the Survey User Settings dialog box (page 205) |
| GradingElevEditor | Change elevations using the Grading Elevation Editor panarama vista (page 755) |
| EditFeatureElevs | Edits feature-line or lot-line elevations using the command line (page 757) |

| Command | Description |
|---|---|
| SetFeatureGrade | Specifies the grade or slope between selected points on a feature line (page 758) |
| InsertElevPoint | Inserts an elevation point (page 765) |
| DeleteElevPoint | Deletes feature line elevation points (page 767) |
| InsertFeatureHighLowPoint | Inserts high or low elevation points into feature lines (page 767) |
| SetFeatureRefElev | Sets the feature line elevation in relation to an existing elevation in the drawing (page 759) |
| RaiseLowerFeaturesByRef | Raise/lower a feature line based on a grade, slope, or relative elevation from a reference point (page 761) |
| AdjacentFeatureElevsByRef | Set the elevation based on a grade, slope, or elevation difference from another feature (page 762) |
| FeatureGradeExtensionByRef | Extends a grading by reference (page 763) |
| RaiseLowerFeatures | Raises or lowers feature lines by elevation (page 764) |
| FeatureElevsFromSurf | Creates feature line elevations from a surface (page 768) |
| InsertFeaturePI | Inserts points of intersection (page 769) |
| DeleteFeaturePI | Deletes points of intersection (page 771) |
| JoinFeatures | Joins multiple feature lines into a single object (page 771) |
| ReverseFeature | Reverses the direction of feature lines for stationing purposes (page 772) |
| EditFeatureCurve | Edit feature line curves (page 773) |
| FilletFeature | Rounds the corners of feature lines (page 776) |
| FitCurveFeature | Converts tessellated curves to true arcs for better grading results (page 777) |
| OffsetFeature | Offsets feature lines at an elevation difference (page 783) |
| AddFeatureLineLabels | Displays the Add Line labels dialog box (page 1970) |
| AddSegmentLabel | Add segment labels to figures (page 359) |
| FeatureLineSegmentLabels | Label feature line segments (page 787) |

| Command | Description |
|---|---|
| AddParcelLineTable | Inserts a parcel line table (page 1565) |
| AddParcelCurveTable | Inserts a parcel curve table (page 1565) |
| AddParcelSegmentTable | Inserts a parcel segment table (page 1565) |
| EditParcelTagNumbers | Activates the renumber label tags command (page 1544) |
| Mapcheck | Displays the Mapcheck Analysis Window (page 1607) |
| OST | Opens the Toolspace Survey tab (page 98) |
| SurveyCalculator | Displays the Astronomic Direction Calculator (page 389) |
| TranslateSvDatabase | Displays the Translate Survey Database dialog box (page 198) |
| ImportSurveyData | Displays the Import Survey Data Wizard (page 2435) |
| EditSvPointProperties | Displays the Survey Point Properties dialog box (page 2416) |
| EditSvFigureProperties | Displays the Survey Figure Properites dialog box (page 2411) |

# Survey Command Language Commands

You can use the survey command language to enter information directly at the Survey Command Window input line.

---

**TIP** In the Survey Command Window, enter HELP at the command line to display a text file that lists all the Survey Command Language commands.

---

For more information, see Survey Command Window (page 2419).

## Command Language Syntax Conventions

When using the survey command language, you enter the command name and data using a specific syntax.

The following conventions are used to present the survey command language syntax:

- Command name: Shown in uppercase.

- []: Square brackets enclose a required parameter, such as a coordinate or an angle measurement.

- ( ): Parentheses enclose an optional parameter, such as a description or point number.

---

**NOTE** Do not insert any brackets or parentheses when you type the command. The only exception to this rule is when you use mathematical operations.

---

**Example**

The following is an example of the syntax to create a point by angle, distance, and vertical angle.

AD VA (point) [angle] [distance] [vertical angle] (description)

The following is an example of the actual data you would enter:

AD VA 5 67.5757 100.63 91.1546 IPF

**NOTE** If you use numeric descriptions such as 100, or descriptions that include one or more spaces such as VERY LARGE MONUMENT, then you should enclose them in quotation marks.

# Point Creation Commands

The following table lists the point creation-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
|---|---|
| AD (point) [angle] [distance] (description) | Creates a point by angle and distance (page 266) |
| AD VA (point) [angle] [distance] [vertical angle] (description) | Creates a point by angle, distance, and vertical angle (page 266) |
| AD VD (point) [angle] [distance] [vertical distance] (description) | Creates a point by angle, distance, and vertical distance (page 266) |
| AP ON (point) | Turns automatic point numbering on (page 264) |
| AP OFF | Turns automatic point numbering off (page 264) |
| BD (point) [bearing] [quadrant] [distance] (description) | Creates a point by bearing and distance (page 268) |
| BD VA (point) [bearing] [quadrant] [distance] [vertical angle] (description) | Creates a point by bearing and distance (page 268) |
| BD VD (point) [bearing] [quadrant] [distance] [vertical distance] (description) | Creates a point by bearing, distance, and vertical distance (page 268) |
| DD (point) [angle] [distance] [vertical angle] (description) | Creates a point by deflection angle and distance (page 270) |
| DD VA (point) [angle] [distance] [vertical angle] (description) | Creates a point by deflection angle, distance, and vertical angle (page 270) |
| DD VD (point) [angle] [distance] [vertical distance] (description) | Creates a point by deflection angle, distance, and vertical distance (page 270) |

| Command | Description |
| --- | --- |
| F1 (point) [angle] [distance] (description) | Creates a point using a FACE 1 angle (page 272) |
| F1 VA (point) [angle] [distance] [vertical angle] (description) | Creates a point using a FACE 1 angle and a vertical angle (page 272) |
| F1 VD (point) [angle] [distance] [vertical distance] (description) | Creates a point using a FACE 1 angle and a vertical distance (page 272) |
| F2 (point) [angle] [distance] (description) | Creates a point using a FACE 2 angle (page 272) |
| F2 VA (point) [angle] [distance] [vertical angle] (description) | Creates a point using a FACE 2 angle and a vertical angle (page 272) |
| F2 VD (point) [angle] [distance] [vertical distance] (description) | Creates a point using a FACE 2 angle and a vertical distance (page 272) |
| LAT LONG (point) [latitude] [longitude] (description) | Creates a control point using a latitude and longitude (page 253) |
| NE (point) [northing] [easting] (description) | Creates a control point using a northing and easting (page 253) |
| NE SS (point) [northing] [easting] (elevation) (description) | Creates a non control point (page 256) |
| NEZ (point) [northing] [easting] (elevation) (description) | Creates a control point using a northing, easting, and elevation (page 253) |
| PRISM [height] | Changes the current prism height |
| PT OFFSET (point) [offset] (ahead) (description) | Createss a point by offsetting the last point (page 276) |
| SKIP | Skips over the next available point (when automatic point numbering is on) |
| STADIA (point) [angle] [distance] [rod] (vertical angle) (description) | Creates a point using the stadia method (page 277) |
| ZD (point) [azimuth] [distance] (description) | Creates a point using azimuth and distance (page 275) |
| ZD [VA] (point) [azimuth] [distance] [vertical angle] (description) | Creates a point using azimuth, distance, and vertical angle (page 275) |
| ZD [VD] (point) [azimuth] [distance] [vertical distance] (description) | Creates a point using azimuth, distance, and vertical distance (page 275) |

# Point Location Commands

The following table lists the point location-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| AZ [point 1] [point 2] [azimuth] | Determines the azimuth between two points (page 378) |
| B [point1] [point2] [bearing] [quadrant] | Defines a bearing between two points (page 377) |
| BS [point] (orientation) | Locates a point by backsight |
| STN [point] (instrument. height) (description) | Creates a new setup station (page 259) |

# Point Information Commands

The following table lists the point information-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| A [point 1] [point 2] [point 3] | Lists the horizontal angle between points (page 376) |
| AZ [point 1] [point 2] | Lists the azimuth between two points (page 378) |
| B [point 1] [point 2] | Lists a bearing between two points (page 377) |
| D [point 1] [point 2] | Determines the distance between two points (page 379) |
| DISP PTS (point 1) (point 2) | Displays point information (page 376) |
| GRADE [point 1] [point 2] | Lists the grade between two points (page 385) |
| INV PTS [point 1] [point 2] | Lists the direction and distance between points (page 380) |
| INVERSE RADIAL [backsight] [station] | Lists the angle and distance from a setup in a radial direction from the backsight point (page 382) |
| SLOPE [point 1] [point 2] | Lists the slope angle between two points (page 387) |

| Command | Description |
| --- | --- |
| SD [point 1] [point 2] | Lists the slope distance between two points (page 384) |
| VD [point 1] [point 2] | List the elevation distance between two points (page 383) |

## Point Editing Commands

The following table lists the point information-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| DEL PTS [point 1] (point 2) | Deletes points from the survey databases (page 278) |
| MOD DESC [point 1] [description] | Modifies a point's description (page 279) |
| MOD ELEV [point 1] [elevation] | Modifies the elevation of a point (page 280) |
| MOD ELEV BY [point 1] [amount] | Modifies the elevation of a point by an increment (page 280) |
| MOD ELS [point 1] [point 2] [elevation] | Modifies the elevation of multiple points (page 280) |
| MOD ELS BY [point 1] [point 2] [amount] | Modifies the elevation of a group of points by an increment (page 280) |

## Figure Commands

The following table lists the figure-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| AREA [figure] | Lists the area and perimeter of a figure (page 358) |
| BEG [figure] | Begins a new figure (page 344) |
| C3 | Draws a 3-point curve (page 325) |
| CLOSE | Closes a figure (page 321) |
| CLOSE BLD | Closes a figure (page 321) |

| Command | Description |
| --- | --- |
| CLOSE RECT [offset] | Closes a figure (page 321) |
| CONT [figure] | Continues a figure to add additional vertices to it (page 346) |
| CRV [DELTA, LENGTH, DEFL, MID, TAN, CHORD] [radius] [value] | Creates a curve in the current figure (page 328) |
| DEL FIG [figure] | Deletes a figure from the drawing (page 353) |
| DISP FIGS | Lists all the existing figure names in the current drawing (page 355) |
| END | Indicates the end of a figure (page 352) |
| FIG AD [angle] [distance] | Creates a figure vertex using an angle and distance (page 329) |
| FIG DD [deflection] [distance] | Creates a figure vertex using a deflection angle and distance (page 332) |
| FIG ZD [azimuth] [distance] | Creates a figure vertex using an azimuth and distance (page 331) |
| FIG BD [bearing] [quadrant] [distance] | Creates a figure vertex using a bearing and distance (page 333) |
| FIG NE [northing] [easting] | Creates a figure vertex using known coordinates (page 334) |
| ID FIG | Identifies a figure (page 354) |
| INVERSE FIG [figure] | Displays the exact closure (inverse) information for a figure (page 357) |
| MCS | Starts a multi-point curve within an active figure (page 326) |
| MCE | Ends a multi-point curve within an active figure (page 326) |
| MAPCHECK [figure] | Displays mapcheck information for a figure (page 356) |
| OFFSET [figure] [distance] | Offsets a figure (page 351) |
| PC | Specifies that the next shot is a point of curvature (page 327) |
| POINT [point] | Selects an existing point to be the next vertex in the figure (page 335) |
| RT [distance] | Defines a right turn in a figure (page 336) |

| Command | Description |
|---|---|
| SET (point) | Creates a point at the current location in a figure (page 337) |
| START [figure] | Extends a figure from the start point (page 345) |
| XC ZD (BULB) [radius] [chord azimuth] [chord distance] | Creates a curve using the chord azimuth (page 343) |
| XC BD (BULB) [radius] [chord bearing] [quadrant] [chord distance] | Creates a curve using the chord bearing (page 341) |
| XC AD (BULB) [radius] [chord angle] [chord distance] | Creates a non-tangential curve (page 338) |
| XC DD (BULB) [radius] [chord deflection] [chord distance] | Creates a curve using the chord deflection (page 342) |
| XC C3 [point on curve] [end point] | Creates a curve using a point on the curve (page 339) |
| XC PTS [radius] [radius point] [end point] | Creates the curve radius (page 340) |

## Intersection Commands

The following table lists the intersection-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
|---|---|
| AZAZ [point 1] [azimuth 1] [offset 1] [point 2] [azimuth 2] [offset 2] | Calculates an azimuth/azimuth intersection (page 305) |
| BB [point 1] [bearing 1] [quadrant 1] [offset 1] [point 2] [bearing 2] [quadrant 2] [offset 2] | Calculates a bearing/bearing intersection (page 304) |
| LNLN [point 1] [point 2] [offset 1] [point 3] [point 4] [offset 2] | Calculates a line/line intersection (page 307) |
| ARCAZ [point] [radius] [point 1] [azimuth] [offset] | Creates an arc/azimuth intersection (page 312) |
| ARCB [point] [radius] [point 1] [bearing] [quadrant] [offset] | Calculates an arc/bearing intersection (page 310) |
| ARCLN [point] [radius] [point 1] [point 2] [offset] | Calculates an arc/line intersection (page 314) |

| Command | Description |
| --- | --- |
| ARCARC [point 1] [radius 1] [point 2] [radius 2] | Calculates an arc/arc intersection (page 316) |
| SAVE [NORTH, SOUTH, EAST, WEST, 1, 2, ALL] (point) (description) | Saves the intersection point (page 317) |
| SAVE [NEAR, FAR] (point) [reference point number] (description) | Saves the intersection point (page 317) |
| SAVE PICK (point) (description) | Saves the intersection point (page 317) |
| SQ [reference 1] [reference 2] [reference] | Calculates a square offset (page 308) |

# Equipment Correction Commands

The following table lists the equipment correction-related AutoCAD Civil survey commands and briefly describes their functionality.

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**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

---

| Command | Description |
| --- | --- |
| ANGLES [RIGHT, LEFT] [ZENITH, NADIR, HORIZ] | Changes the horizontal angle type (right or left) and the vertical angle type |
| ATMOS [ON, OFF] | Turns atmospheric corrections on or off |
| COLL [ON, OFF] | Turns the collimation corrections for measurements on or off |
| CR [ON, OFF] | Turns curvature and refraction corrections on or off |
| EDM OFFSET [offset] | Sets the EDM (Electronic Distance Measuring) offset for an instrument |
| HORIZ ANGLE [RIGHT, LEFT] | Changes the horizontal angle type |
| PRESS [pressure] [INCH, MBAR, MM] | Specifies the atmospheric pressure |
| SF [factor] | Changes the scale factor |
| PRISM CONSTANT [constant] | Changes the prism constant |
| PRISM OFFSET [offset] | Changes the prism offset |
| TEMP [temperature] [F, C, K] | Sets the temperature for atmospheric corrections |

| Command | Description |
|---|---|
| UNITS [METER, FOOT] [DMS, DECDEG, GRAD, MILS, RADIANS] | Changes the distance or angle units for measurements |
| VERT ANGLE [ZENITH, NADIR, HORIZ] | Changes the vertical angle reference system |

# Baseline Commands

The following table lists the baseline-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
|---|---|
| BL IS [point 1] [point 2] [station] | Defines a baseline (page 285) |
| BL PT (point 1) [station] (offset) (description) | Creates a point at an offset from a baseline (page 287) |
| BL INV [point 1] (point 2) | Lists a baseline direction using inverse point (page 286) |

# Centerline Commands

The following table lists the centerline-related AutoCAD Civil survey commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
|---|---|
| CL IS [figure] (station) (point) | Defines a centerline (page 289) |
| CL INV [point 1] (point 2) | Displays point data in reference to the current centerline (page 291) |
| CL PT (point) [station] (offset) (skew angle) (description) | Creates a new point by station offset (page 293) |
| CL PT BY [point] [station 1] [offset] [distance] (station 2) (description) | Creates points at an increment along the centerline (page 293) |
| CL ELEV (point) [station] [offset] [elevation] (description) | Creates a point by station, offset, and elevation (page 295) |
| CL ROD (point) [station] [offset] [rod] (description) | Creates a point using station, offset and rod height (page 296) |

| Command | Description |
| --- | --- |
| CL VD (point) [station] [offset] [vertical distance] (description) | Creates a point along a centerline using a vertical distance (page 297) |
| HI [elevation] | Defines the height of instrument (page 292) |
| XS [station] | Sets the current cross section station (page 299) |
| XS ELEV (point) [offset] [elevation] (description) | Creates a point with offset and elevation from the current cross section (page 300) |
| XS ROD (point) [offset] [rod] (description) | Creates a point by offset and rod height from the current cross section (page 301) |
| XS VD (point) [offset] [vertical distance] (description) | Creates a point by offset and vertical distance from the current cross section (page 302) |

## AutoCAD Related Commands

The following table lists the AutoCAD commands that you can use at the AutoCAD Civil survey command line and briefly describes their functionality.

The following table lists the corridor-related AutoCAD Civil commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| PAN | Shifts the location of the drawing |
| REDRAW | Refreshes the display in the current viewport |
| ZOOM PT [point] | Zooms to a point |
| ZOOM [WINDOW, EXTENTS, PREVIOUS, W, P, E, A] | Zooms according to one of the specified parameters |

## Miscellaneous Commands

You can use commands to quickly access survey functionality.

The following table lists the survey-related AutoCAD Civil commands and briefly describes their functionality.

**NOTE** These commands must be entered in the Survey Command Window. For more information, see Survey Command Window (page 2419).

| Command | Description |
| --- | --- |
| CALC [formula] | Performs mathematical calculations |
| DITTO [ON, OFF] | Turns the Ditto feature on or off |

| Command | Description |
| --- | --- |
| HELP (command) | Displays the syntax of the specified command |
| HISTORY | Displays all the commands you used during the current session |
| OUTPUT [ON, OFF] | Turns the use of an output file on or off |
| TRAV [ON, OFF] | Turns the traverse command on or off |

# Point Data

# Points

# 15

Points are basic building blocks in AutoCAD Civil. You can use points in land development projects to identify existing ground locations and design elements.

Points are numbered and named uniquely. Each point has properties that can include information such as northing, easting, elevation, and description. A point that is displayed in a drawing can have additional properties that control its appearance, such as a point style, a point label style, and a layer.

A point can be included in an AutoCAD Civil project so that it can be accessed by multiple users, or it can exist outside a project in a single drawing. For more information about the differences between project and drawing points, see Managing Points (page 423).

Points Workflow

## Understanding Points

In AutoCAD Civil, you can create, work with, and manage points using a variety of commands and tools.

The points created by AutoCAD Civil are called Coordinate Geometry (COGO) points, which are very different from AutoCAD point nodes. AutoCAD point nodes have only coordinate data (XYZ values) associated with them. However, COGO points, in addition to coordinate data, have a variety of properties associated with them, including point number, point name, raw (field) description, and full (expanded) description. Unlike AutoCAD point nodes, which exist in a single drawing, COGO points can be stored in a project outside a drawing and referenced by multiple users. In AutoCAD Civil, the term point refers to a COGO point, not to an AutoCAD point node.

### Displaying and Labeling Points

To control the appearance of a point symbol in a drawing, either select a default point symbol, or create a point symbol. For more information, see Point Styles (page 419).

You can automatically label points with predefined labeling criteria as you create points in a drawing. For more information, see Point Labels and Tables (page 421).

### Editing Points

Use the Point Editor to edit point properties. For more information, see Point Properties (page 408). You can also use AutoCAD commands to edit points graphically. For more information, see Editing Points in a Drawing (page 488).

**Creating Points**

Choose from many point creation commands to create points. For more information, see Creating Points (page 427). You can also create points by importing point data from a file. For more information, see Importing and Exporting Points (page 495).

When you create or import points, you can use description keys to automatically control the appearance of a point in the drawing based on its raw (field) description. For more information, see Description Keys (page 537).

**Managing and Organizing Points**

You can group related points into point groups using a variety of criteria. You can use point groups to control the appearance of points in a drawing, to create surfaces, or to export selected points to a file. For more information, see Point Groups (page 521).

Use a project to manage and protect the points needed for a design project. When you use a project as a central repository for your project points, the points can be viewed and copied by others, but not necessarily modified.

## The Point Object

Points are AutoCAD Civil objects that can be displayed in the drawing and manipulated graphically. Point appearance is controlled using point labels and point styles.

In the early releases of AutoCAD Civil points were a subcomponents of a point group. Now points in AutoCAD Civil are individual objects where you can perform the following:

■ You can select points individually in the drawing and graphically change the location and rotation using grips.

■ You can create a selection set of points to move their elevations to a surface.

■ You can modify point properties in the AutoCAD Properties dialog box which describes both the basic and specific AutoCAD Civil properties.

■ You can change the Next Point Number settings in the point creation commands.

**NOTE** To perform any of these functions, the point must be checked out, unlocked, and not a Survey Point.

Point Groups are still used as a way to organize points and to manage points for selection, overrides, and adding points to surfaces.

When creating points you specify the point layer in the Edit Command Settings-CreatePoints dialog box. For more information, see Edit Point Settings dialog box (page 2103).

You can use the AutoCAD LIST command to display point properties.

## Point Properties

Use the Prospector tree to access point properties.

Point properties specify all the information associated with a point, including its data, such as point number, northing, easting, and elevation. Properties for a drawing point include information about how the point is displayed in a drawing, and properties for a project point include information about its status with respect to the project.

Display the properties of a point, even a point that is not in the current drawing, by clicking either a drawing Points collection or a project Points collection in the Prospector tree to display a list view. For more information, see The Toolspace Item View (page 82).

For a description of all drawing and project point properties, see Point Editor (page 2130). Use the Point Editor to edit drawing point properties. View the properties of project points in the project Points collection list view.

Point group overrides affect how some point properties, including elevation and raw description, are displayed and are used in a drawing without changing the stored value of the point property. You can also use point group overrides to override point styles and point label styles. For more information, see Using Point Groups to Override Point Properties (page 523).

## Points Collection (Prospector Tab)

Use the Points collection in the Prospector tree to access the points in a drawing.

Right-click the drawing Points collection to:

- Create a new point. (page 427)

- Export the points in a drawing. (page 505)

- Transfer points in a file to another file format. (page 506)

- Edit the points in a drawing with the Point Editor. (page 488)

- Zoom or pan to the points in the drawing. (page 426)

- Lock or unlock the points in the drawing. (page 423)

- Refresh the Prospector tree.

**NOTE** The Points shortcut menu in the Prospector tree and in the Points list view can also contain project management access control menu items when they are available. The menu items that display can change depending on the status of the selected point relative to the project, such as whether it is checked out.

If one or more points have been added to the current drawing, you can view a list of the points in the list view next to or beneath the Prospector tree. For more information, see The Toolspace Item View (page 82).

## Point Collection (Settings Tab)

Use the Point collection in the Settings tree to manage point settings, point styles, point label styles, point file formats, description key sets, table styles, and point command settings.

Right-click the Point collection to:

- Edit the point feature settings. (page 410)

- Edit the point styles defaults. (page 419)

- Refresh the Settings tree.

Expand the Point collection to display and edit the styles, command settings, and other collections available for points, which are listed in the following table:

| For more information about... | Follow this link... |
|---|---|
| Point Styles | The Object Style Collection (Settings Tree) (page 94) |
| Label Styles | The Label Styles Collections (Settings Tree) (page 95) |
| Point File Formats | Point File Formats Collection (Settings Tab) (page 496) |
| Description Key Sets | Description Key Sets Collection (Settings Tree) (page 538) |
| External Data References | Using External Data References (page 508) |
| User-Defined Property Classifications | User-Defined Property Classifications (page 416) |
| Table Styles | The Table Styles Collections (Settings Tree) (page 96) |
| Commands | Point Settings (page 410) |

# Point Settings

Point settings control how point-related commands behave.

You work with point settings the same way you work with other object settings in AutoCAD Civil, using the Toolspace Settings tree. You can control point-related settings at both the object collection (feature) level and at the command level. For information about how the levels of settings work together, see Understanding Settings (page 61).

Use the Settings tree Point collection shortcut menu to establish defaults for all point-specific settings and to override the drawing ambient settings for all point-related commands. Use the Commands collection under the Settings tree Point collection to override point-specific settings or drawing ambient settings for a specific command.

**NOTE** Overrides to the drawing ambient settings at the Point collection level and the Point Commands collection level affect only the specified level. The drawing level settings are not affected. For more information, see Understanding Settings (page 61).

The topics in this section describe only those settings that affect point-related commands. They do not describe the drawing ambient settings that you can edit at the Point collection level and the Point Commands collection level, even though those settings are displayed. For more information about the drawing ambient settings, see Specifying Ambient Settings (page 68).

## Editing the Default Styles Settings

Before creating points, specify the default point style and point label style used to create the points if description keys are not used.

The styles you specify establish the default settings for the styles in the Create Points dialog box. When you select the Create Points command, you can expand the Create Points dialog box and edit these default settings.

---

**IMPORTANT** By default, in the Create Points dialog box - Points Creation section the Disable Description Keys value is set to false and the controls in the Default Styles section are not available. To enable the Default Styles controls, set the Disable Description Keys value to true.

---

**To edit the Default Style settings**

1 Open the Edit Point Settings (page 2103) dialog box using one of the following methods:
  - Edit settings for all point-related commands: In Toolspace, on the Settings tab, right-click Point ➤ Edit Feature Settings.
  - Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools ⊕. Expand the Create Points dialog box. Expand Points Creation.
  - Edit settings for a specific command: In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the desired command. Click Edit Command Settings.

2 If needed, scroll to Default Styles and expand the collection.

3 To specify a default point style, edit the Point Style setting.

4 To specify a default point label style, edit the Point Label Style setting.

5 Click OK.

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools ⊕. Expand the Create Points dialog box. Expand Points Creation.

Toolspace Shortcut Menu

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Point Commands collection ➤ right-click <command> ➤ Edit Command Settings

Command Line

ShowPointSettings

Dialog Box

Point Settings (page 2103)

# Editing the Default Name Format

Before creating points, specify the default name template for points and point groups.

**To edit the Default Name Format**

1 Open the Edit Point Settings (page 2103) dialog box using one of the following methods:
  - Edit settings for all point-related commands: In Toolspace, on the Settings tab, right-click Point ➤ Edit Feature Settings.

- Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools . Expand the Create Points dialog box. Expand Points Creation.

- Edit settings for a specific command: In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the desired command. Click Edit Command Settings.

2 If needed, scroll to Default Name Format and expand the collection.

3 To specify a default Point Group Name Template, click in the Point Group Name Templatevalue column and enter values in the Name Template dialog box.

4 To specify a default Point Name Template, click in the Point Name Template value column and enter values in the Name Template dialog box.

5 Click OK.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools . Expand the Create Points dialog box. Expand Points Creation.

Toolspace Shortcut Menu

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Points ➤ Commands collection ➤ right-click <command> ➤ Edit Command Settings

Command Line

ShowPointSettings

Dialog Box

Point Settings (page 2103)

## Editing the Update Points Settings

Use the Update Points setting to control whether you can change project points without checking them out.

Normally, after you check a point in to a project, you cannot modify the point in the drawing. To modify it, you must first check it out of the project. In general, you should not modify a project point unless you first check it out of the project.

However, there may be times when you want to modify a point that you have not checked out. In that case, use the Update Points setting, Allow Checked-In Points To Be Modified.

**WARNING** If you edit points without checking them out, you cannot check the points in to the project. Your changes exist only in the drawing in which you made them.

**To edit the Update Points settings**

1 Open the Edit Point Settings (page 2103) dialog box using one of the following methods:

- Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools . Expand the Create Points dialog box. Expand Points Creation.

- Edit settings for all point-related commands: In Toolspace, on the Settings tab, right-click the Point ➤ Edit Feature Settings.

- Edit settings for a specific command: In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the desired command and click Edit Command Settings.

**2** If needed, scroll to Update Points and expand the collection.

**3** Edit the Allow Checked In Points To Be Modified setting.

**4** Click OK.

## Quick Reference

**Ribbon**

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools ⚙ . Expand the Create Points dialog box. Expand Points Creation.

**Toolspace Shortcut Menu**

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Point Commands collection ➤ right-click <command> ➤ Edit Command Settings

**Command Line**

ShowPointSettings

**Dialog Box**

Point Settings (page 2103)

# Editing the Point Identity Settings

Before creating or importing points, specify the settings that control how the points are numbered and how point number collisions are handled.

If you import points from an external file, and the points in your point data file do not have point numbers, use the Point Identity settings to control how the points are numbered as they are imported.

If you import points from an external file, and the points in your point data file have point numbers, specify how point number conflicts are resolved if the point file contains a point number that already exists. The following example shows how the If Point Numbers Already Exists settings resolves conflicts when your point data file contains a point number that already exists:

**Example: If you have a point in the drawing with the following properties:**

- Number: 23

- Northing: 500

- Easting: 500

- Elevation: 70.5

- Description: IP

You import a point data file containing a point with the same point number, but different properties, as follows:

- Number: 23

- Northing: 502.18

- Easting: 498.65

If you set the If Point Number Already Exists setting to Overwrite before you import the point file, the point in the drawing becomes:

Number: 23 Northing: 502.18 Easting: 498.65 Elevation: blank Description: blank

All properties are overwritten, even properties that aren't defined in the point data file, such as elevation and description.

If you set the If Point Number Already Exists setting to Merge before you import the point file, the point in the drawing becomes:

Number: 23 Northing: 502.18 Easting: 498.65 Elevation: 70.5 Description: IP

The number, northing, and elevation are overwritten, but the elevation and description in the drawing are preserved.

**To edit the Point Identity settings**

1 Open the Edit Point Settings (page 2103) dialog box using one of the following methods:

- Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools ⊕ . Expand the Create Points dialog box. Expand Points Creation.

- Edit settings for a specific command: In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the desired command and click Edit Command Settings.

2 If needed, scroll the dialog box to Point Identity and expand the collection.

3 To specify point names and point numbers and to resolve duplicate point names and point numbers, edit the following settings:

- Next Point Number

- Use Sequential Numbering

- Point Number Offset

- Sequence Point Numbers From

- If Point Numbers Are Supplied

- Force Names

- If Point Numbers Already Exist

- If Point Names Already Exist

- If Point Numbers Need To Be Assigned

4 Click OK.

**Quick Reference**

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 . Expand the Create Points dialog box. Expand Points Creation.

Toolspace Shortcut Menu

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Point Commands collection ➤ right-click <command> ➤ Edit Command Settings

Command Line

ShowPointSettings

Dialog Box

Point Settings (page 2103)

## Editing the Default Point Creation Settings

Use the point creation settings to change the command entry prompts, point elevations and description key options.

The settings you specify when you right-click CreatePoints ➤ Edit Command Settings establish the default settings in the Create Points dialog box. When you select the Create Points command, you can expand the Create Points dialog box and edit these default settings.

**To edit the Point Creation settings**

1   Open the Edit Point Settings (page 2103) dialog box using one of the following methods:

■ Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 . Expand the Create Points dialog box. Expand Points Creation.

■ In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the CreatePoints ➤ Edit Command Settings.

2   If needed, scroll to Points Creation and expand the collection.

3   Specify the settings.

4   Click OK.

**Quick Reference**

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 . Expand the Create Points dialog box. Expand Points Creation.

Toolspace Shortcut Menu

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Point Commands collection ➤ right-click CreatePoints ➤ Edit Command Settings

Command Line

ShowPointSettings

Dialog Box

[Point Settings](page 2103) (page 2103)

## Editing the Default Layer Settings

Before creating points, specify the default layer for the points you create.

**To edit the Default Layer settings**

1 Open the Edit Point Settings (page 2103) dialog box using one of the following methods:

■ Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools  . Expand the Create Points dialog box. Expand Points Creation.

■ In Toolspace, on the Settings tab, expand the Commands collection under the Point collection. Right-click the CreatePoints ➤ Edit Command Settings.

2 If needed, scroll to Default Layer.

3 To specify a new Layer, click the Value cell and specify a layer in the Layer Selection dialog box.

4 Click OK.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools  . Expand the Create Points dialog box. Expand Points Creation.

Toolspace Shortcut Menu

Settings tab: right-click Point collection ➤ Edit Feature Settings
OR
Settings tab: Point Commands collection ➤ right-click <command> ➤ Edit Command Settings

Command Line

ShowPointSettings

Dialog Box

[Point Settings](page 2103) (page 2103)

## User-Defined Property Classifications

You can use User-Defined Property Classifications to organize and group specific types of user-defined properties.

With user-defined properties, you can assign, label, edit, and query point data. User-defined properties are grouped together in User-Defined Property Classifications. For example, if you want to group all user-defined properties for points representing trees, start by creating a classification called Trees. This classification is displayed as a collection in User-Defined Property Classifications on the Settings tab of Toolspace. Then create and add properties that are specific to the classification. For example, in the Tree Classification some

properties you might create are name, species, and height. Each property you create is displayed in the Toolspace list view.

# Creating a User-Defined Property Classification

Create a User-Defined Property Classification that contains a common group of properties.

**To create a user-defined property classification**

1  On the Settings tab in Toolspace, expand the object Collection.

2  Right-click User-Defined Property Classifications. Click New.

3  In the User-Defined Property Classification dialog box, enter a name for the classification.

4  Click OK.

The new classification is displayed under User-Defined Property Classifications on the Settings tab of Toolspace and also in the list view window.

## Quick Reference

Toolspace Shortcut Menu

Settings tab ➤ Point collection ➤ right-click User-defined Property Classification ➤ New

# Creating a User-Defined Property

Add user-defined properties to User-Defined Property Classifications collection.

**To create a user-defined property**

1  On the Settings tab in Toolspace, expand the object folder.

2  Expand the User-Defined Property Classifications folder.

3  Right click a classification. Click New.

4  In the New User-defined Property Dialog Box (page 1811) dialog box, enter a name, description, and property field type.

5  Click OK.

The new property is displayed in Toolspace and also in the list view window.

## Quick Reference

Toolspace Shortcut Menu

Settings tab ➤ Point collection ➤ right-click a named User-defined Property Classification ➤ New

# Editing, Copying, or Deleting User-Defined Properties

Use the shortcut menu in the Toolspace list view to make edits to user- defined properties.

**To edit a user-defined property**

**1** On the Settings tab in Toolspace, click a User-defined Classification.

**2** In the list view window, right-click a property cell. Select one of the following:

- Edit Enumeration. For more information, see the Edit Enumeration (page 1810) dialog box.

- Delete

- Line Shading

- Copy to Clipboard

### Quick Reference

List View Shortcut menu

Right-click a user-defined property cell ➤ Edit Enumeration, Delete, Line Shading, or Copy to Clipboard

## Assigning User-Defined Properties to Point Groups

Assign a User-Defined Classification to a point group.

After you create a User-Defined Classification, you can assign it to a point group using the Classification column of the Point Groups item list view. The properties within the classification are displayed as columns in the Point Group item list view.

**To assign user-defined properties to a point group**

**1** In the drawing, define the properties within the User-Defined Classification. For more information, see Creating a User-Defined Property Classification (page 417).

**2** In Toolspace, on the Prospector tab, click the Point Groups collection.

**3** In the Point Groups list view window, click the Classification column. Select the User-Defined Classification you want to assign to the specific point group.

**4** In Toolspace, on the Prospector tab, select the point group to which you assigned the User-Defined Classification.

**5** In the point group list view window, right-click a column heading. Select the user-defined properties you want to display in the list view window.

> **NOTE** You can arrange the display of columns in the list view by dragging column headings. For more information, see Customizing a List View (page 84).

## Importing and Exporting User-Defined Property Values

Import and export User-defined property values using Point File Formats and the Import and Export commands.

When you compose a Point File Format, the user-defined property names are displayed in the Point File Formats - Select Column (page 2122) dialog box. You can select a property and assign it to a specific column.

**To import user-defined property values into the drawing and assign them to points**

**1** In the drawing, define the properties within the User-Defined Classification.

**2**  Define a point file format that contains columns with the user-defined properties. The point file must have a column containing the point identity, either Name or Number, that identifies which point is assigned the user-defined property.

**3**  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🛠️ .

**4**  Click the Import Points button.

**5**  Specify information in the Import Points dialog box. For more information, see Importing Point Data (page 503).

**To export user-defined properties**

**1**  Click Output tab ➤ Export panel ➤ Export Points 🛠️ .

**2**  Specify settings in the Export Points dialog box. For more information, see Exporting Point Data (page 505).

## Quick Reference

Ribbon

Click Output tab ➤ Export panel ➤ Export Points 🛠️

Menu

Click Points menu ➤ Import/Export Points ➤ Export Points.

Command Line

ExportPoints

# Point Styles

Point styles control how point symbols display in a drawing.

Manage point styles the same way you manage all object styles in AutoCAD Civil, using the Toolspace Settings tree. In the Settings tree, all objects have a standard object style grouping called an object style collection, which you use to create, edit, copy, and delete the styles for that object. For more information, see The Object Style Collection (Settings Tree) (page 94).

When defining a point style, select from a set of default point symbols, or create a point symbol using an AutoCAD Block. For more information about creating an AutoCAD Block, see AutoCAD Help.

Assign a point style to a drawing point either when you create the point or import the point. Use either the Prospector Points list view or the Point Editor to change the point style. The point style referenced by a point is not necessarily the point style used to display the point in the drawing. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

# Overview of Point Styles

Use styles to control the appearance of a point symbols and a point labels in a drawing.

A point can directly reference two styles:

■  Point Style. Defines how a point symbol is displayed in the drawing. To change a point symbol, either edit the point style or change the point so that it references a different point style. For more information, see Point Styles (page 419).

- Point Label Style. Defines how a point is labeled in the drawing. To change the way a point is labeled, you can either edit the point label style or change the point so that it references a different point label style. For more information, see Point Labels and Tables (page 421).

The point style determines the appearance of the point symbol, and the point label style determines how the point is labeled:
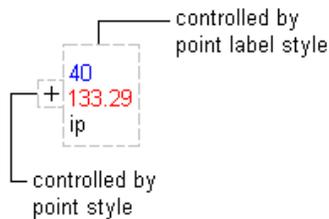


Although an individual point can reference a point style and a point label style, when possible, you will want to use point groups to control the display of point symbols and point labels for points sharing similar characteristics. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Creating a Point Style

Use the Settings tree to create a point style.

A point style specifies the symbol that displays at the X Y location for the point in the drawing. The style can also specify scaling for the symbol and its appearance in 3D views.

**To create a point style**

1. In Toolspace, on the Settings tab, right-click the Point Styles collection ➤ New.

2. In the Point Style dialog box, click the Information (page 2114) tab. Enter a name and description for the point style.

3. To define the symbol used to display the point, click the Marker (page 2115) tab. Specify the symbol type and options for size, scaling, and rotation.

4. To define how the point displays in 3D views, click the 3D Geometry (page 2116) tab. Specify the 3D Geometry settings.

5. To define the display properties for the point style, click the Display (page 2117) tab. Specify the display properties for the symbol.

6. To view summary information about the style, click the Summary (page 2117) tab.

7. Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Point Style collection ➤ New

Dialog Box

Point Style (page 2114)

# Creating a Point Style Based on an Existing Point Style

Use the Settings tree to create a point style based on an existing point style.

Create a point style from an existing point style by copying an existing point style and then editing the copy.

**To create a point style from an existing point style**

1  In Toolspace, on the Settings tab, right-click the point style you want to copy. Click Copy.

    A copy is created and its properties are displayed in the Point Style dialog box.

2  In the Point Style dialog box, click the Information (page 2114) tab. Enter a name and description for the new point style.

3  Use the tabs in the Point Style (page 2114) dialog box to modify the point style's properties. For more information, see Creating a Point Style (page 420).

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click Point Style item  ➤  Copy

Dialog Box

  Point Style (page 2114)

# Editing a Point Style

Use the Settings tree to edit a point style.

**To edit a point style**

1  In Toolspace, on the Settings tab, right-click the desired point style. Click Edit.

2  In the Point Style (page 2114) dialog box, edit the properties of the point style.

    For more information, see Creating a Point Style (page 420).

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click Point Style item  ➤  Edit

Dialog Box

  Point Style (page 2114)

# Point Labels and Tables

Label and table styles control the appearance and behavior of point labels and tables in a drawing.

This section describes characteristics unique to point labels and point tables. For general information about labels and tables, see Labels and Tags (page 1463) or Tables (page 1555).

**Point Labels**

Assign a point label style to a drawing point either when you create the point or import the point. Use the Points list view on the Prospector tab or the Point Editor to change the style. The point style referenced by a point is not necessarily the point style used to display the point in the drawing. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Point Tables**

When you insert a point table into a drawing, specified point information is automatically displayed in the table.

## Point Label Styles

Use the Settings tree to manage and create point label styles.

You can create multiple label styles for point objects.

You can label points when you create them, or you can use point groups and description keys to specify how points are labeled in a drawing. You can also edit text on individual point labels.

For information about changing existing point label styles, see Modifying Labels in a Drawing (page 1533).

**To edit point label style defaults**

1   On the Settings tab in Toolspace, expand the Points collection.

2   Click the Label Styles collection, right click ➤ Edit Label Style Defaults.

For more information, see Adding Text Components to Labels (page 1503).

**To create a point label style**

1   On the Settings tab in Toolspace, expand the Points collection.

2   Click the Label Styles collection, right click ➤ New. For more information, see Creating and Editing Label Styles. (page 1477)

For more information, see Adding Text Components to Labels (page 1503).

**To edit point label text**

1   In the drawing, right-click a point ➤ Edit Label Text.

2   At the command line you are prompted to select a text component.

3   In the Text Component Editor dialog box, make the necessary changes.

4   Click OK.

For more information, see Adding Text Components to Labels (page 1503).

## Point Tables

Use Point tables to display information about points in a drawing.

Specify the points to include in the table by selecting them from the drawing or specifying a point group. The point information is automatically displayed in the table when you insert the point table into the drawing.

Unlike some objects, that have multiple table styles, points have only the point table style.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. The procedure in this topic explains how to access the point table creation command. It provides a link to information about the Point Table Creation dialog box. For information about modifying tables, see Modifying Tables (page 1567).

**To create a point table**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables Menu ➤ Add Point Table ⊞ .

2   In the Point Table Creation (page 2135) dialog box, change the generic table settings as needed.

3   Click 🔷 to select a point group that specifies the points to be included in the table.

4   Click ↖ to select points in the drawing to be included in the table.

5   Click OK.

6   Select the location for the upper-left corner of the table in the drawing.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables Menu ➤ Add Point Table ⊞

Menu

Click Points menu ➤ Add Tables.

Command Line

AddPointTable

# Managing Points

You can manage points using an AutoCAD Civil project.

For example, you can add points to a project and protect them so they can be viewed and copied by others, but not modified. For general information about project management, see Project Management (page 125). For information about working with points in a project management environment, see Working with Vault Project Points (page 166).

**NOTE** Survey points that are created using the Prospector Survey tab are not included in project management operations even if they are selected. For example, any Survey points included in the selection list during the Add to Project command are not added to the project.

You can populate a project with points by creating the points in the drawing and adding them to the project. For more information, see Creating a Project Point Database.

For a recommended procedure for checking out, editing, and checking in a large number of project points, see Modifying A Large Number of Project Points (page 494).

# Locking and Unlocking Points

Lock a point to prevent its properties from being modified in a drawing.

A locked point is designated by  in the Prospector tree and the Point collection list view. The icon is displayed only if the Toolspace tree drawing state icons are displayed. For more information, see Drawing Item State Icons (page 141).

**NOTE** A locked point is locked only in the drawing. A protected point is protected at the project level. For more information, see Protecting Project Points (page 171).

A locked point cannot have any of its properties changed, including its point style and point label style. It cannot be edited, deleted, or overwritten. However, you can reposition its label in the drawing.

When you are working with a drawing point that is not related to a project point, lock the point to prevent changes as previously described. Unlock the point to change it, including changing its XYZ values.

When you work with a local copy of a project point or Survey point, the locking behavior of a point changes. When you check a drawing point into a project, the local copy of the point is automatically locked, so that no changes can be made to it as described previously.

If you unlock a point with either a project state of checked in, checked out to other, or protected, you cannot modify its XYZ values, but you can change its point styles.

If you need to change a local copy of a project point in a drawing, use the Allow Checked In Points To Be Modified setting. For more information, see Editing the Update Points Settings (page 412). It is important to remember that any changes you make to the point can never be checked in to the project.

**WARNING** If you edit points without checking them out, you cannot check the points in to the project. Your changes exist only in the drawing in which you made them.

Lock or unlock all the points in a drawing using the Points collection shortcut menu. Lock or unlock individual drawing points using the list view shortcut menu. Lock or unlock all the points in a point group point list using a single command.

**To lock or unlock points**

1  In Toolspace, on the Prospector tab, click the Points collection to display the Points list view.

2  Select the desired points. For more information, see Selecting Items in a List View (page 84).

3  Right-click then click Lock or Unlock.

## Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Lock Points 
or

Click COGO Point tab ➤ Modify panel ➤ Unlock Points 

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Lock or Unlock

Command LIne

LockPoints
or
UnlockPoints

# Controlling the Appearance of Points in a Drawing

Several factors control how a point displays in a drawing.

**Point Style and Point Label Style**

The point style and the point label style control the basic appearance of a point symbol and label in a drawing.

The point style determines the appearance of the point symbol, and the point label style determines how the point is labeled:
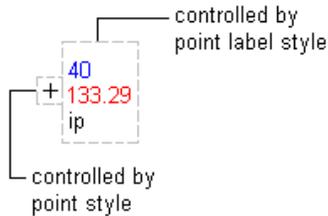


Each individual drawing point can directly reference a point style and a point label style. These *individual point styles* are assigned when a point is created. The individual point styles assigned during point creation are determined either by description key matching or by the styles specified in the Create Points dialog box. A point is not required to reference a point style or point label style. (If a point does not reference a point style, the style referenced by the layer on which the point is created is used.

If a point does not reference a point style or point label style, the point style or point label style property for the point is empty when viewed in the Point Editor or the Points list view. You can change or remove individual point styles using the Point Editor or Points list view.

A point group can reference a default point style and a default point label style. View or change these point group default styles using the Information tab (page 2108) or the Overrides tab (page 2113) on the Point Group Properties dialog box. If you change the style on one tab, the other tab updates to reflect the change.

In addition, the default styles for a point group can be designated as point group override styles. Use the check box on the Overrides tab (page 2113) of the Point Group Properties dialog box to specify that a point group style is an override style.

The following describes, in order of precedence, which of the previously described styles are used to display a point in a drawing:

**Style**

| | |
|---|---|
| Point Group Override Styles | The Object Style Collection (Settings Tree) (page 94)<br>If the point belongs to more than one point group that specifies override styles, the override styles specified by the highest point group in the point group display order are used to draw the point. For more information, see Changing the Point Group Display Order (page 526). |
| Individual Point Styles | The Label Styles Collections (Settings Tree) (page 95) |
| Point Group Default Styles | Point File Formats Collection (Settings Tab) (page 496) |
| The _All Points group | Description Key Sets Collection (Settings Tree) (page 538) |

**NOTE** All points belong to the _All Points point group. To display all the points in a drawing that use the same style, specify an override style for the _All Points point group, and move it to the top of the point group display order.

**Point Style Display Settings**

Use the Point Style display settings to control whether the point symbol and/or the point label is drawn for any point that is displayed using the point style. On the Display tab in the Point Style dialog box, the Visible column controls the visibility of point symbols and point labels. To change the visibility of point symbols, change the visibility for the Marker component. To change the visibility of point labels, change the visibility for the Label component.

**Layers**

Use the point layer to control the appearance of a point in a drawing.

Specify a point layer for a point by setting its Layer property. Set the Layer property either during point creation (using description key matching or the options in the Create Points dialog box) or after a point is created (using the Point Editor or the Points list view).

If a point belongs to more than one point group, the highest point group controls the visibility of the point in the point group display order.

A point can be displayed using the properties of a layer, such as visibility, color, line type, and line weight. If an individual point style or point label style is used to display a point (as described earlier in this topic), the ByBlock or ByLayer specifications in the point style or point label style refer to the layer that is assigned to the point.

# Printing Points

The Copy To Clipboard command lets you copy a list of drawing or project points to another application for printing.

To print a list of points in a drawing or project, click the drawing or project Prospector tree collection to display the points in a list view. Right-click the list view and select Copy To Clipboard to copy the list view into a file you can print. For information, see Copying Items from a List View (page 86).

# Zooming and Panning to Points

Use the list view shortcut menu to automatically zoom or pan to points in the drawing.

**To zoom or pan to points**

**1**  In Toolspace, on the Prospector tab, click the drawing Points collection.

**2**  In the list view, select the points you want to zoom or pan to. For more information, see Selecting Items in a List View (page 84).

**3**  Right-click and click Zoom To or Pan To.

**Quick Reference**

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Zoom To, Pan To

# Creating Points

You can use the Create Points dialog box to create points using a variety of methods.

For standardization, you can use a drawing template that contains point settings, point styles, and point label styles.

In the Create Points dialog box, choose commands from the following lists:

■ Miscellaneous

■ Intersection

■ Alignments

■ Surface

■ Interpolate

■ Slope

■ Import Points

You can expand the Create Points dialog box to access the settings that are relevant to point creation. If you change the settings in this dialog box the changes are reflected in the CreatePoints command settings. For more information, see Edit Feature Settings - Point Dialog Box (page 2103).

You can also use the Civil transparent commands in combination with the Create Points Manual command to create points using known information such as angle and distance. For more information about the Civil transparent commands, see Transparent Commands (page 1581).

You can use description key substitution while creating drawing points to automatically assign point style, point label style, layer, full description, and scale and rotation of the point symbol. For more information, see Understanding Description Keys (page 537).

## Before You Create Points

Save time and effort by considering certain issues before creating points.

The following factors can affect both the point creation process and the way you work with points after they are created:

■ When creating drawing points, work in a drawing based on a template which contains the point settings, point styles, point label styles, and description keys you want to use when creating the point. For more information, see Drawing Templates (page 109).

■ Plan how you will control the point style and point label style used to draw points in a drawing. You can set the point styles at the point group level, or at the point level, and automatically assign this method of style selection during point creation. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

■ If you plan to use description keys to assign styles, layers, and other information when creating drawing points, make sure that the description keys are present in the drawing before you create the points. For more information, see Understanding Description Keys (page 537).

■ Specify settings that control, among other things, how you are prompted during the point creation commands and how the created points are numbered. For more information, see Editing the Default Point Creation Settings (page 415).

**NOTE** If prompted for an elevation or description and you want to create a point without these, enter a period (.) at the prompt.

# Creating Points Using Miscellaneous Methods

Use the icons in the Miscellaneous list to access commands that are commonly used for creating points.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in a drawing. For more information, see Before You Create Points (page 427).

## Creating Points at Specified Coordinates

Creates a point at specified or selected location in the drawing.

Enter X, Y (and Z, if prompted) coordinates at the command line, or click a location in the drawing.

For more information, see Specifying Locations in a Drawing (page 115)

**To create points at specified coordinates**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Click Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Miscellaneous ➤ Manual ⊕.

> **NOTE** You can also use transparent commands to specify point locations within a larger operation, such as the creation of an alignment or a parcel lot line. Using transparent commands, you can calculate the location for a point from information such as angle and distance, or from point object information, such as a point number. For more information, see Transparent Commands (page 1581).

3 Specify the location for the point in the drawing. For more information, see Specifying Locations in a Drawing (page 115).

4 If prompted, enter the point name, description, and elevation.

5 Optionally, repeat steps 4 and 5.

6 Press Enter to end the command.

> **NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Manual ⊕

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Manual

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointManual

Dialog Box

Create Points (page 2117)

# Creating Points by Geodetic Direction and Distance

Creates a point by using a specified geodetic direction.

From the starting point, enter the geodetic azimuth and the geodesic distance to the location where you want to create the point. To use this command, you must assign a coordinate system to the drawing.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in a drawing. For more information, see Before You Create Points (page 427).

**To create points by geodetic direction and distance**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Set the zone and transformation settings for the drawing.

3 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Geodetic Direct ⊕ .

4 Specify a starting point.

5 Enter the geodetic azimuth from the point specified in Step 5 to the location where you want to create the point.

6 Enter the geodesic distance, which is the distance measured from the point specified in Step 5 to the location where you want to create the point.

7 If prompted, enter the point name, description, and elevation.

8 Optionally, repeat steps 5-8.

9 Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Geodetic Direct ⊕

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Geodetic Direct

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePoints

Dialog Box

Create Points (page 2117)

## Creating Points by Resection

Creates a point at a position calculated from the measured angles between three known points.

To create the point, enter the backsight (reference) point, then enter the two points that are sighted on, followed by the angles for each of the sighted points.
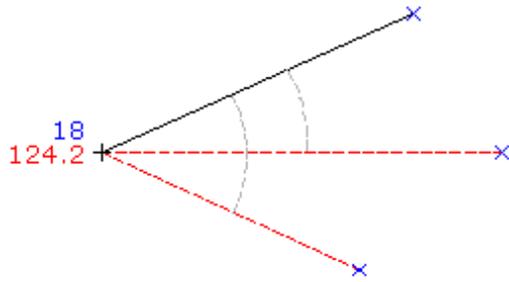


**To create points by resection**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Resection .

3   Specify the first point, which is the backsight (or reference) point.

4   Specify the second point.

5   Specify the third point.

6   Enter the angle between the first point and the second point.

7   Enter the angle between the first point and the third point.

8   If prompted, enter the point name, description, and elevation.

9   Optionally, repeat steps 4-9.

10   Press Enter to end the command.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Resection

Menu

   Points menu ➤ Create Points - Miscellaneous ➤ Resection

Toolspace Shortcut Menu

   Prospector tab: right-click Points collection ➤ Create

Command Line

   CreatePointResection

Dialog Box

   Create Points (page 2117)

## Creating Points by Station and Offset

Creates a point at a specified station and offset distance from a line, polyline, feature line, lot line, or arc.

Use the station and the offset to specify the location of the point.
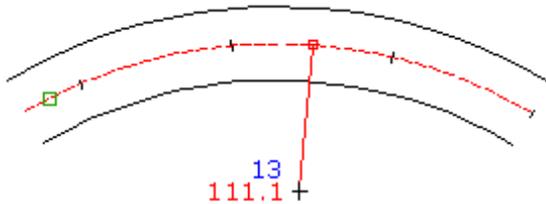


**To create points by station and offset**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Station/Offset Object  .

3   Select a line, polyline, feature line, lot line, or arc.

4   Enter the starting station or press Enter to accept the default.
    The ending station is automatically calculated.

5   Enter the station at which you want to create the point or press Enter to accept the default.

6   Enter the offset distance.

7   If prompted, enter the point name, description, and elevation.

8   Optionally, repeat steps 4-8.

9   Press Enter to end the command.

---

**NOTE**  Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Station/Offset Object 

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Station/Offset Object

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtStationOffsetObj

Dialog Box

Create Points (page 2117)


## Creating Points at Object Vertices and Critical Geometric Points

Creates points automatically at the endpoints of lines, feature lines, or lot lines, or at the endpoints and center point of arcs.

Points are created from a selection set of objects. Points are not created where lines cross unless one of the points is an end point.
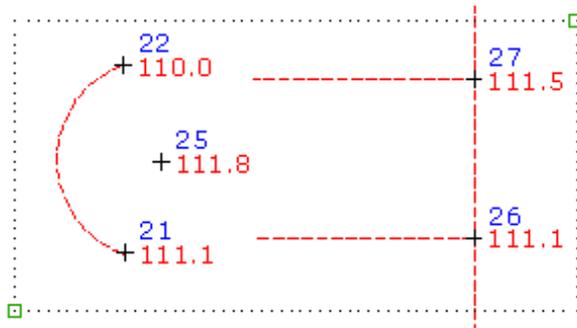


To create a point where lines cross each other, use the Create Points Manual command.

**To create points at object vertices and critical geometric points**

1　Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2　Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Automatic .

3　Select objects. You can select lines, feature lines, lot lines, and arcs.

4　A point is placed at each vertex of the selected object.

5　For each point, if prompted, enter the point name, description, and elevation.

6　Optionally, repeat steps 4-5.

**7** Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

**Quick Reference**

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Automatic ⛛

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Automatic

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointAutomatic

Dialog Box

Create Points (page 2117)

## Creating Points Along a Line or a Curve

Creates points along a line, feature line, lot line, or arc at a specified distance from an end point.

Click near the end point from which you want to measure the distance and enter a distance to specify the location for the created point along the object.
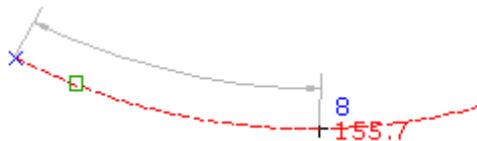


**To create points along a line or a curve**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Along Line/Curve ⛛ .

**3** Select a line, lot line, feature line, or arc.

The end point nearest to the point you selected is highlighted.

**4** Enter the distance from the end point.

**5** If prompted, enter the point name, description, and elevation.

**6** Optionally, do one of the following:

■ Enter another distance from the start point of the selected object and follow the prompts as previously described.

■ Press Enter, select another object, and follow the prompts as previously described.

**7** Press Enter twice to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Along Line/Curve

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Along Line/Curve

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtAlongLnCurveSpir

Dialog Box

Create Points (page 2117)

## Creating Points on a Line or Curve

Creates a point at the endpoint of a line, feature line, lot line, or arc and on points of intersection (PIs) and radius points for arcs.

Locations of the specified points are created for an arc. Select one object at a time. If you select intersecting objects, duplicate points are not created.
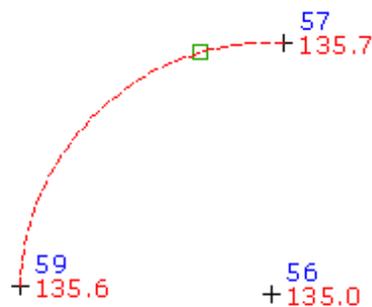
When you create points using this command, you work with one object at a time. After points have been added using a selected object, you can select another object and add points using that object. If the objects you select and the options you specify while using the command would result in the creation of duplicate points, for example, at the shared endpoints of two lines, only a single point is added. Duplicate points are not created.

**To create points on a line or curve**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ On Line/Curve .

3 Select a line, lot line, feature line, or arc.

4 Points are placed at geometric points on the selected object.

5 For each point, if prompted, enter the point name, description, and elevation.

6 Optionally, repeat steps 4-5.

7 Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ On Line/Curve

Menu

Points menu ➤ Create Points - Miscellaneous ➤ On Line/Curve

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtOnLineCurveSpiral

Dialog Box

Create Points (page 2117)

## Creating a Specific Number of Points Along an Object by Dividing the Object

Creates a specific number of evenly spaced points along a line, feature line, lot line, or arc.

Specify the number of segments, and a point is placed at each vertex. Points can placed on the object or at an offset.
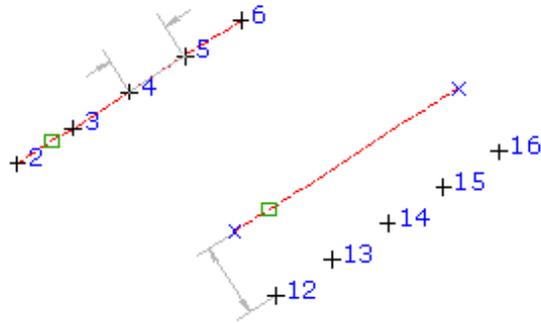
When you create points using this command, you work with one object at a time. After points have been added using a selected object, you can select another object and add points using that object. If the objects you select and the options you specify while using the command would result in the creation of duplicate points, for example, at the shared endpoints of two lines, only a single point is added. Duplicate points are not created.

**To create a specific number of points along an object by dividing the object**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Divide Object .

3  Select a line, lot line, feature line, or arc.

4  Enter the number of segments to divide the object into.

5  Enter the offset.
   A series of equally spaced points are placed along the object at the specified offset.

6  For each point, if prompted, enter the point name, description, and elevation.

7  Optionally, repeat steps 4-7.

8  Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Divide Object

Menu

   Points menu ➤ Create Points - Miscellaneous ➤ Divide Object

Toolspace Shortcut Menu

   Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDivideObject

Dialog Box

Create Points (page 2117)

## Creating Points on an Object Using Distance Intervals

Creates points that are a specified distance apart along a line, feature line, lot line, or arc.

Specify the start and end stations on the object, distance (interval) between points, and optional offset.

Use a specified interval and offset to place points along an object.
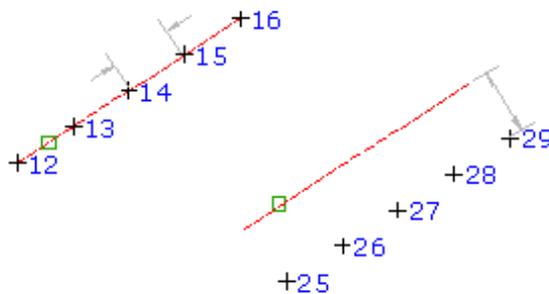


When you create points using this command, you work with one object at a time. After points have been added using a selected object, you can select another object and add points using that object. If the objects you select and the options you specify while using the command would result in the creation of duplicate points, for example, at the shared endpoints of two lines, only a single point is added. Duplicate points are not created.

**To create points on an object using distance intervals**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Measure Object ⟨icon⟩ .

3   Select a line, lot line, feature line, or arc.

4   Enter the starting station or press Enter to accept the default.

5   Enter the ending station or press Enter to accept the default.

6   Enter the offset.

7   Enter the distance interval.

    Points are placed along the object at the specified offset and interval.

8   For each point, if prompted, enter the point name, description, and elevation.

9   Optionally, repeat steps 4-9.

10  Press Enter to end the command.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Measure Object

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Measure Object

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointMeasureObject

Dialog Box

Create Points (page 2117)

## Creating Points on Polylines at a Specified Elevation

Creates points at the vertices of a polyline at a specified elevation.

Enter the elevation and then select the polyline. Points are placed at the vertices at the specified elevation.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in a drawing. For more information, see Before You Create Points (page 427).

**To create points on polylines at a specified elevation**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Manual .

3  Enter the elevation.

4  Select a polyline.
   Points are placed at the vertices of the polyline at the specified elevation.

5  For each point, if prompted, enter the point name and description.

6  Optionally, repeat steps 4-6.

7  Press Enter to end the command.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Quick Reference**

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Manual ⊕

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Manual

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtPlylineCtrVertMan

Dialog Box

Create Points (page 2117)

## Creating Points on Polylines

Creates points at the vertices of a polyline.

Draw a polyline with elevations, and then use this command to create points along the polyline.

**To create points on polylines**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Automatic ⊕ .

3  Select a polyline.
   Points are placed at the vertices of the polyline.

4  For each point, if prompted, enter the point name and description.

5  Optionally, repeat steps 4-6.

6  Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

**Quick Reference**

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Automatic ⊕

Menu

Points menu ➤ Create Points - Miscellaneous ➤ Polyline Vertices - Automatic

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtPlylnCtrVertAuto

Dialog Box

[Create Points](page 2117)

## Creating Points at Intersections

Use the icons in the Intersection list to access commands that you can use for creating points at intersections.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in a drawing. For more information, see Before You Create Points (page 427).

### Creating Points at a Direction/Direction Intersection

Creates a point at the intersection of two directions, each defined by a point and a bearing or azimuth.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.

Create a point at the intersection by specifying the offset distance from each direction line.
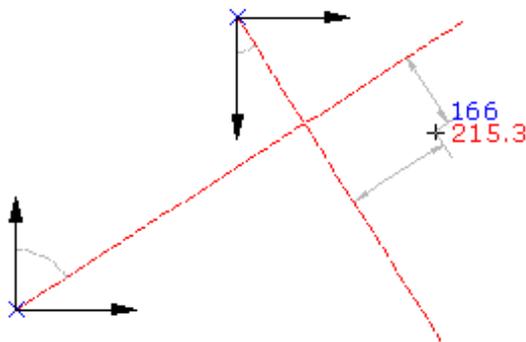


**To create a point at a direction/direction intersection**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Direction ⬚ .

3 Specify the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

4 Specify the direction from the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the second point.

■ Enter Z. Enter the azimuth of the direction line.

■ Enter B. Specify the quadrant and the bearing.

**5** Specify the offset distance by doing one of the following:
■ Enter a distance.

■ Pick a point in the drawing to define the offset.

■ Use Transparent commands to specify a point to define the offset.

**6** Repeat steps 4-6 to define the second direction line.

**7** Enter a description, or press Enter to skip the description.

**8** Enter an elevation for the point, or press Enter to skip the elevation.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Direction
⬇⊕

Menu

Points menu ➤ Create Points - Intersections ➤ Direction/Direction

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDirectionDir

Dialog Box

Create Points (page 2117)

## Creating Points at a Distance/Distance Intersection

Creates a point at the intersection of two distances, each defined by a radius length.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.
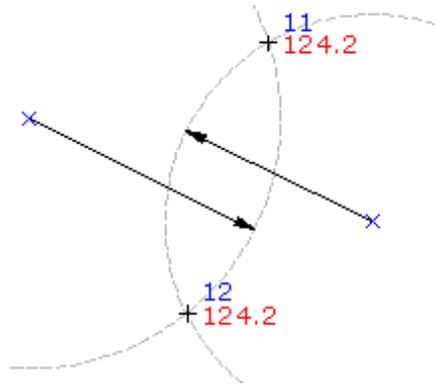
**To create points at a distance/distance intersection**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Distance 🔯 .

3 Specify the radial point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the radial point.

4 Specify the radius by doing one of the following:
   ■ Enter a radius value.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the radius.

5 Repeat steps 5-6 to specify a location for the radial point.

   In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

6 Specify the intersection point by doing one of the following:
   ■ Click near the X on which you want to create the point.
   ■ Enter A to place points at both intersections.

7 Enter a description, or press Enter to skip the description.

8 Enter an elevation for the point, or press Enter to skip the elevation.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Distance 🔯

Menu

Points menu ➤ Create Points - Intersections ➤ Distance/Distance

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDistanceDist

Dialog Box

Create Points (page 2117)

## Creating Points at a Direction/Distance Intersection

Creates a point at the intersection of a direction (line) and distance (circle).

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.
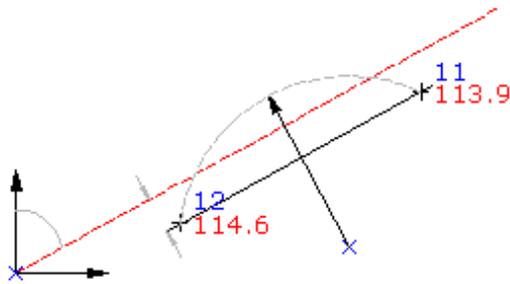


**To create points at a direction/distance intersection**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Distance ⤧.

3  Specify the radial point by doing one of the following:
   ■  Pick a point in the drawing.

   ■  Use Transparent commands to specify the point.

4  Specify the radius by doing one of the following:
   ■  Enter a radius value.

   ■  Pick a point in the drawing.

   ■  Use Transparent commands to specify the radius.

5  Specify the start point by doing one of the following:
   ■  Pick a point in the drawing.

   ■  Use Transparent commands to specify the point.

**6** Specify the direction from the start point by doing one of the following:

■ Pick a point in the drawing.

■ Use Transparent commands to define the point.

■ Enter Z, and then enter the azimuth of the direction line.

■ Enter B, and then specify the quadrant and the bearing.

**7** Specify the offset distance by doing one of the following:

■ Enter a distance.

■ Pick a point in the drawing.

■ Use Transparent commands to specify a point to define the offset.

In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

**8** Specify the intersection point by doing one of the following:

■ Click near the X on which you want to create the point.

■ Enter A to place points at both intersections.

**9** Enter a description, or press Enter to skip the description.

**10** Enter an elevation for the point, or press Enter to skip the elevation.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Distance
⬐⊕

Menu

Points menu ➤ Create Points - Intersections ➤ Direction/Distance

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDirectionDist

Dialog Box

Create Points (page 2117)

## Creating Points at a Direction/Perpendicular Intersection

Creates a point that is on a direction line and perpendicular to a selected point.

Select a point in the drawing to create a point on the direction line that is perpendicular to the selected point.
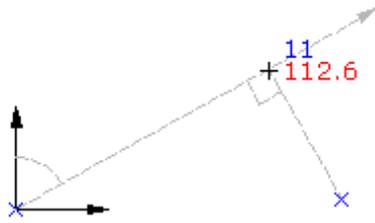
**To create points at a direction/perpendicular intersection**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Perpendicular .

3  Specify the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

4  Specify the direction from the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to define the point.
   ■ Enter Z, and then enter the azimuth of the direction line.
   ■ Enter B, and then specify the quadrant and the bearing.

5  Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

6  Specify the perpendicular point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the perpendicular point.

7  Enter a description, or press Enter to skip the description.

8  Enter an elevation for the point, or press Enter to skip the elevation.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

## Creating Points at a Distance/Perpendicular Intersection

Creates a point that is radial to both a circle and a selected point.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.
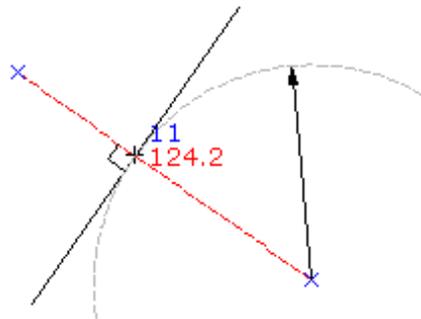


**To create points a distance/perpendicular intersection**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Perpendicular .

3   Specify the radial point by doing one of the following:
    ■ Pick a point in the drawing.

    ■ Use Transparent commands to specify the point.

4   Specify the radius by doing one of the following:
    ■ Enter a radius value.

    ■ Pick a point in the drawing.

■ Use Transparent commands to specify the radius.

**5** Enter a description, or press Enter to skip the description.

**6** Enter an elevation for the point, or press Enter to skip the elevation.

**7** Specify the perpendicular point by doing one of the following:

■ Pick a point in the drawing.

■ Use Transparent commands to specify the perpendicular point.

---

**8** **NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

**9** Enter a description, or press Enter to skip the description.

**10** Enter an elevation for the point, or press Enter to skip the elevation.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points -

Intersections ➤ Distance/Perpendicular

Menu

Points menu ➤ Create Points - Intersections ➤ Distance/Perpendicular

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDistancePerp

Dialog Box

Create Points (page 2117)

## Creating Points at a Direction/Object Intersection

Creates a point at the intersection of a line, curve, or spiral object and a direction line.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.
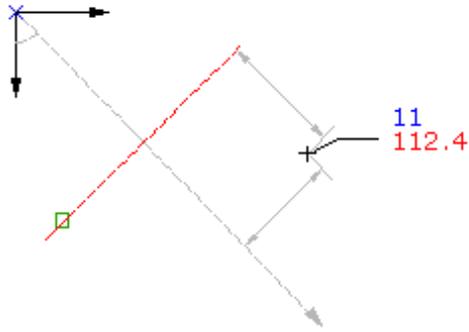
**To create points at a direction/object intersection**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Intersections ➤ Direction/Object .

**3** Select an object.

**4** Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

**5** Specify the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

**6** Specify the direction from the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to define the point.
   ■ Enter Z, and then enter the azimuth of the direction line.
   ■ Enter B, and then specify the quadrant and the bearing.

**7** Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

**8** Enter a description, or press Enter to skip the description.

**9** Enter an elevation for the point, or press Enter to skip the elevation.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Quick Reference**

## Creating Points at a Distance/Object Intersection

Creates a point at the intersection of an object and a radial distance.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line. Points are offset from the intersection of an object and a radial distance.
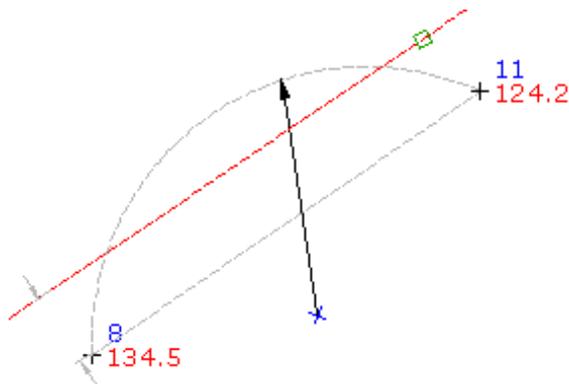


**To create points at a distance/object intersection**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Object ↕⊕ .

3   Select an object.

4   Specify the offset distance by doing one of the following:

   ■  Enter a distance.

   ■  Pick a point in the drawing.

   ■  Use Transparent commands to specify a point to define the offset.

**5** Specify the radial point by doing one of the following:

- Pick a point in the drawing.

- Use Transparent commands to specify the point.

**6** Specify the radius by doing one of the following:

- Enter a radius value.

- Pick a point in the drawing.

- Use Transparent commands to specify the radius.

In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

**7** Specify the intersection point by doing one of the following:

- Click near the X on which you want to create the point.

- Enter A to place points at both intersections.

**8** Enter a description, or press Enter to skip the description.

**9** Enter an elevation for the point, or press Enter to skip the elevation.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Object
↕⊕

Menu

Points menu ➤ Create Points - Intersections ➤ Distance/Object

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDistanceObject

Dialog Box

Create Points (page 2117)

## Creating Points at an Object/Object Intersection

Creates a point at the intersection of lines, arcs, lot lines, or feature lines.

If the objects do not intersect, the command extends either object to determine the intersection point. A point is offset from the intersection of two objects.
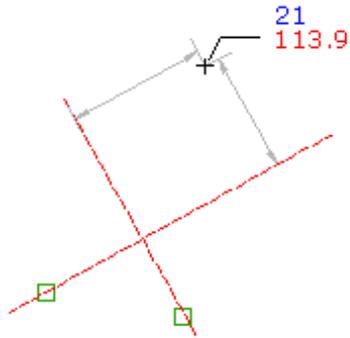
**To create points at an object/object intersection**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Object/Object 🔶 .

3  Select an object.

4  Specify the offset distance by doing one of the following:
   ■  Enter a distance.
   ■  Pick a point in the drawing.
   ■  Use Transparent commands to specify a point to define the offset.

5  Select the second object.

6  Specify the offset distance by doing one of the following:
   ■  Enter a distance.
   ■  Pick a point in the drawing.
   ■  Use Transparent commands to specify a point to define the offset.

   In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

7  Specify the intersection point by doing one of the following:
   ■  Click near the X on which you want to create the point.
   ■  Enter A to place points at both intersections.

8  Enter a description, or press Enter to skip the description.

9  Enter an elevation for the point, or press Enter to skip the elevation.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Quick Reference**

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Object/Object

Menu

Points menu ➤ Create Points - Interpolate ➤ Intersection

Toolspace Shortcut Menu

Prospector tab: Right-click Points collection ➤ Create

Command Line

CreatePointObjectObject

Dialog Box

Create Points (page 2117)

## Creating Points at a Perpendicular Intersection

Creates a point along a line or arc object, perpendicular, or radial to a specified point.

From a specified point, the created point is perpendicular on a line but radial on a curve. The point is perpendicular to a selected point and a line.
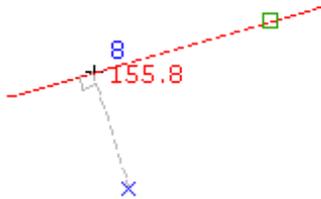


**To create points at a perpendicular intersection**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Perpendicular .

3 Select an object.

4 Specify the perpendicular point by doing one of the following:
   ■ Pick a point in the drawing.

   ■ Use Transparent commands to specify the perpendicular point.

5 Enter a description, or press Enter to skip the description.

**6** Enter an elevation for the point, or press Enter to skip the elevation.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

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## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Perpendicular

Menu

Points ➤ Create Points - Intersections ➤ Perpendicular

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtInterpPerpendic

Dialog Box

Create Points (page 2117)

## Creating Points at a Direction/Alignment Intersection

Creates a point either at the intersection of a direction line and an alignment or at a point offset from the intersection.

Specify values by picking points in the drawing, using transparent commands, or entering values at the command line.

Create a point that is offset from the intersection of a direction and an alignment by selecting a direction line and then specifying the offset from the direction line and the offset from the alignment.



**To create points at a direction/alignment intersection**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Alignment ↓⊕ .

**3** Select an alignment object.

**4** Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

**5** Specify the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

**6** Specify the direction from the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to define the point.
   ■ Enter Z, and then enter the azimuth of the direction line.
   ■ Enter B, and then specify the quadrant and the bearing.

**7** Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

**8** Specify the intersection point by doing one of the following:
   ■ Click near the X on which you want to create the point.
   ■ Enter A to place points at both intersections.

**9** Enter a description, or press Enter to skip the description.

**10** Enter an elevation for the point, or press Enter to skip the elevation.

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**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

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## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Direction/Alignment ↓⊕

Menu

Points menu ➤ Create Points - Intersections ➤ Direction/Alignment

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDirectionAlign

Dialog Box

Create Points (page 2117)

## Creating Points at a Distance/Alignment Intersection

Creates a point at the intersection of a distance line and an alignment.

The intersection is located where the radial distance from a specified point crosses the alignment.
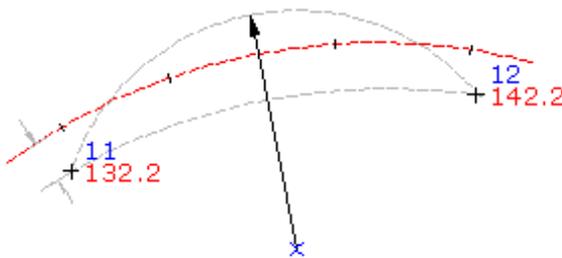


**To create points at distance/alignment intersection**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Intersections ➤ Distance/Alignment ↑⊕ .

3   Select an alignment object.

4   Specify the offset distance by doing one of the following:
    ■  Enter a distance.
    ■  Pick a point in the drawing.
    ■  Use Transparent commands to specify a point to define the offset.

5   Specify the radial point by doing one of the following:
    ■  Pick a point in the drawing.
    ■  Use Transparent commands to specify the point.

    In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

6   Specify the radius by doing one of the following:
    ■  Enter a distance.
    ■  Pick a point in the drawing.

- ■ Use Transparent commands to specify a point to define the offset.

7  Specify the intersection point by doing one of the following:
   - ■ Click near the X on which you want to create the point.

   - ■ Enter A to place points at both intersections.

8  Enter a description, or press Enter to skip the description.

9  Enter an elevation for the point, or press Enter to skip the elevation.

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**NOTE**  Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

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## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Distance/Alignment

Menu

Points menu ➤ Create Points - Intersections ➤ Distance/Alignment

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointDistanceAlign

Dialog Box

Create Points (page 2117)

## Creating Points at an Object/Alignment Intersection

Creates a point at either the intersection of any object and an alignment or at an offset from the intersection.

If the line or arc object does not intersect the alignment, the command extends either object to determine the intersection point.
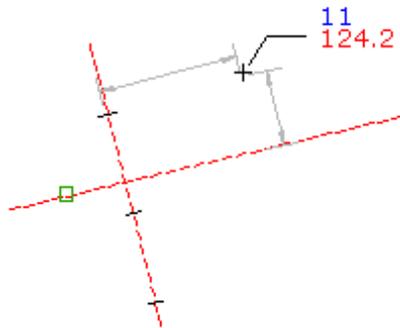
**To create points at an object/alignment intersection**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Intersections ➤ Object/Alignment ⟨icon⟩ .

3 Select an alignment object.

4 Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

5 Select an object.

6 Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

   In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

7 Specify the intersection point by doing one of the following:
   ■ Click near the X on which you want to create the point.
   ■ Enter A to place points at both intersections.

8 Enter a description, or press Enter to skip the description.

9 Enter an elevation for the point, or press Enter to skip the elevation.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Object/Alignment ⟨icon⟩

Menu

Points menu ➤ Create Points - Intersections ➤ Object/Alignment

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointObjectAlign

Dialog Box

Create Points (page 2117)

## Creating Points at an Alignment/Alignment Intersection

Creates a point either at the intersection of two alignments or an offset distance from one or both alignments.

There must be an alignment intersection in the drawing. Pick two intersecting alignments to create a point that is offset from the intersection.
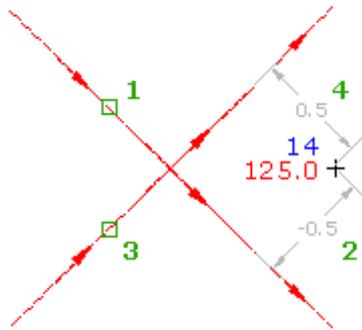


**To create points at an alignment/alignment intersection**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Alignment/Alignment ⊕ .

3   Select an alignment object.

4   Specify the offset distance by doing one of the following:
     ■   Enter a distance.
     ■   Pick a point in the drawing.
     ■   Use Transparent commands to specify a point to define the offset.

5   Select a second alignment object.

6   Specify the offset distance by doing one of the following:
     ■   Enter a distance.
     ■   Pick a point in the drawing.
     ■   Use Transparent commands to specify a point to define the offset.

     In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

7   Specify the intersection point by doing one of the following:
     ■   Click near the X on which you want to create the point.
     ■   Enter A to place points at both intersections.

8   Enter a description, or press Enter to skip the description.

**9** Enter an elevation for the point, or press Enter to skip the elevation.

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**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

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### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Intersections ➤ Alignment/Alignment ⌖

Menu

Points menu ➤ Create Points - Intersections ➤ Alignment/Alignment

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointAlignAlign

Dialog Box

Create Points (page 2117)

## Creating Points Based on Horizontal Alignments

Use the icons in the Alignment list to access commands that you can use to create points relative to a horizontal alignment.

Create points either on a horizontal alignment or at an offset from the alignment.

If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in a drawing. For more information, see Before You Create Points (page 427).

## Creating Points Offset from Alignment Stations

Creates that are offset from stations on an alignment.

Use this command to create ROWs, shoulders, and passing lanes.
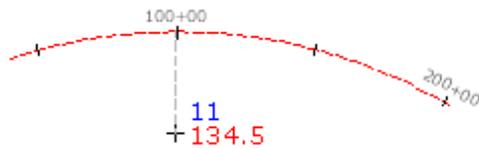
**To create points offset from alignment stations**

**1**  Choose default settings, style, layers, point group, and description keys. For more information, see Before You Create Points (page 427).

**2**  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Station/Offset  .

**3**  Select an alignment.

> **NOTE** If the selected alignment has associated profiles, the Select a Profile dialog box displays. You can select a profile from the list and the elevations for the points you create will be set from that profile. If there is no profile section for a station on the alignment, or if you select <none>, the point elevation will be set according to the Points Creation settings in the Create Points dialog box.

**4**  Enter the station number from which you want to offset a point. Do not include the station plus (+) sign.

**5**  Enter the offset. If you have selected a profile, the offset value for the profile displays as the default.

**6**  If prompted, enter the name, description, and elevation for the point.

> **NOTE**  If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

**7**  Optionally, repeat steps 4-7.

**8**  Press Enter to end the command.

> **NOTE**  Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Station/Offset 

Menu

Points menu ➤ Create Points - Alignments ➤ Station/Offset

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointStationOffset

Dialog Box

Create Points (page 2117)

## Creating a Specific Number of Points Along an Alignment

Creates points that are a specified distance apart along an alignment.

Specify the number of segments for the alignment. Points can be placed at each vertex on the alignment or at a specified offset.
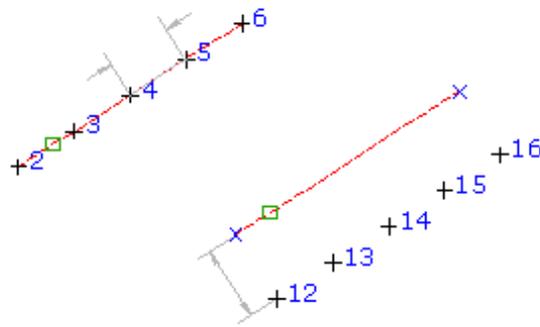


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create a specific number of points along an alignment**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Divide Alignment 🔯 .

3   Select an alignment.

> **NOTE**  If the selected alignment has associated profiles, the Select a Profile dialog box displays. You can select a profile from the list and the elevations for the points you create will be set from that profile. If there is no profile section for a station on the alignment, or if you select <none>, the point elevation will be set according to the Points Creation settings in the Create Points dialog box.

4   Enter the number of segments into which you want to divide the selected alignment.

The command calculates the distances between points by dividing the total length of the alignment by the number of segments specified.

5   Enter the offset. If you have selected a profile, the offset value for the profile displays as the default.

A series of equally spaced points are placed along the alignment at the specified offset.

6   If prompted, for each point, enter a point name, description, and elevation.

**NOTE** If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

**TIP** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

Ribbon

  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Divide Alignment

Menu

  Points menu ➤ Create Points - Alignments ➤ Divide Alignment

Toolspace Shortcut Menu

  Prospector tab: right-click Points collection ➤ Create

Command Line

  CreatePointDivideAlign

Dialog Box

  Create Points (page 2117)

## Creating Points on an Alignment Using a Distance Interval

Creates points at equally spaced intervals along an alignment.

Specify the distance between points, and optionally an offset distance from the alignment.
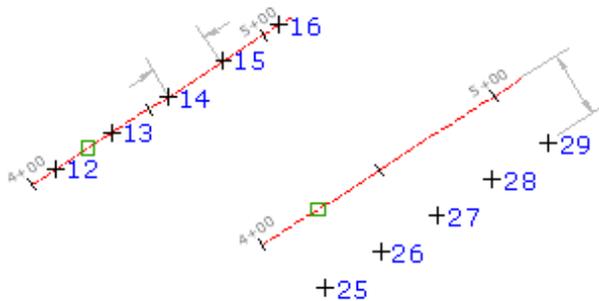


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points on an alignment based on a distance interval**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Measure Alignment ⚘ .

**3** Select an alignment.

> **NOTE** If the selected alignment has associated profiles, the Select a Profile dialog box displays. You can select a profile from the list and the elevations for the points you create will be set from that profile. If there is no profile section for a station on the alignment, or if you select <none>, the point elevation will be set according to the Points Creation settings in the Create Points dialog box.

**4** Enter the starting station or press Enter to accept the default.

**5** Enter the ending station or press Enter to accept the default.

**6** Enter the offset. If you have selected a profile, the offset value for the profile displays as the default.

**7** Enter the interval.

Points are placed along the alignment at the specified offset and interval.

**8** If prompted, for each point, enter a point name, description, and elevation.

> **NOTE** If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

> **TIP** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Measure Alignment ⚘

Menu

Points menu ➤ Create Points - Alignments ➤ Measure Alignment

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointMeasureAlig

Dialog Box

Create Points (page 2117)

## Creating Points at the Geometry Points of Alignments

Creates a point at every geometry point on an alignment.

Geometry points are points of curvature (PC), points of tangency (PT), spiral-curves (SC), curve-spirals (CS), tangent spirals (TS), spiral-tangents (ST), and points of intersection (PI).
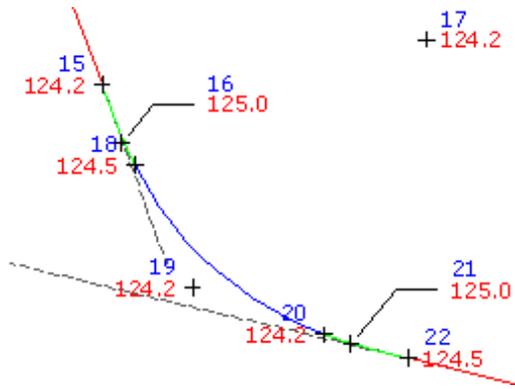
When creating points using this command, you are not prompted for a description if the Prompt For Descriptions Points Creation setting is set to Automatic-Object. The raw and full description for the point is automatically assigned based on the type of point created. The assigned description displays in the drawing if the current point label style contains a raw description or a full description text component.

The following table describes alignment vertex labels that are created and the description that is assigned based on the point type:

| Created Label | Definition |
| --- | --- |
| BOA | Beginning of Alignment |
| PI | Point of Intersection |
| CPI | Curve Point of Intersection |
| PT | Point of Tangency |
| PC | Point of Curvature |
| SPI | Spiral Point of Intersection |
| TS | Tangent-Spiral intersection |
| CS | Curve-Spiral intersection |
| ST | Spiral-Tangent intersection |
| CC | Curve Center or radius point |
| EOA | End of Alignment |

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at the geometry points of alignments**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ At Alignment Geometry ⊕ .

3 Select an alignment.

**NOTE** If the selected alignment has associated profiles, the Select a Profile dialog box displays. You can select a profile from the list and the elevations for the points you create will be set from that profile. If there is no profile section for a station on the alignment, or if you select <none>, the point elevation will be set according to the Points Creation settings in the Create Points dialog box.

**4** Enter the starting station or press Enter to accept the default.

**5** Enter the ending station or press Enter to accept the default.

Points are placed along the alignment object.

**6** If prompted, for each point, enter a point name and elevation.

The created points are automatically labeled using the current label style. If the current label style contains either a raw description or a full description text component, the vertex type displays in the label.

**NOTE** If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ At Alignment Geometry 

Menu

Points menu ➤ Create Points - Alignments ➤ At Geometry Points

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointAtPtPcScetc

Dialog Box

Create Points (page 2117)

## Creating Radial or Perpendicular Points on Alignments

Creates a point on an alignment that is radial or perpendicular to a specified point.

From specified point, the created point is perpendicular on a line and radial on a curve.
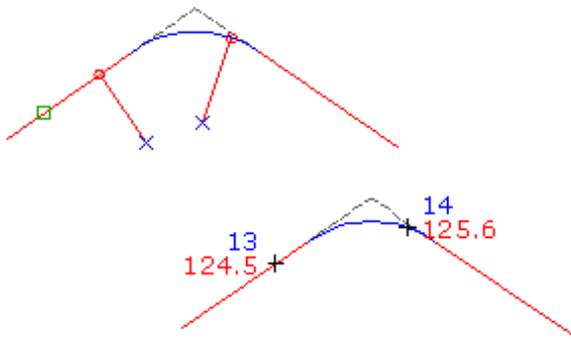
Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create radial or perpendicular points on alignments**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Radial Or Perpendicular 🔬 .

3  Select an alignment.

> **NOTE** If the selected alignment has associated profiles, the Select a Profile dialog box displays. You can select a profile from the list and the elevations for the points you create will be set from that profile. If there is no profile section for a station on the alignment, or if you select <none>, the point elevation will be set according to the Points Creation settings in the Create Points dialog box.

4  Specify a point.

5  If prompted, enter the point name, description, and elevation.

> **NOTE** If you want the point description to be taken from the object, specify Automatic - Object in the Prompt For Descriptions section of the Points Creation settings. The alignment name and station display in the Raw Description column in the Point Editor.

6  Optionally, repeat steps 4-6.

7  Press Enter to end the command.

> **NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Radial Or Perpendicular 🔬

Menu

Points menu ➤ Create Points - Alignments ➤ Radial Or Perpendicular

Command Line

CreatePointRadialOrPerp

Dialog Box

Create Points (page 2117)

## Creating Profile Geometry Points on Alignments

Creates points on the alignment at the geometry points of the profile view with elevations from the profile.

Profile geometry points can be points for the begin of vertical curvature (BVC), points of vertical intersection (PVI), and points of end of vertical curvature (EVC).

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create profile geometry points on alignments**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Profile Geometry Points 📍 .

3  Select an alignment in the drawing, or if there is more than one alignment, select it from the list.

4  Select a Profile from the list.

   Points are created on the alignment at the geometry points of the profile view with elevations from the profile.

5  Press Enter to end the command.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Profile Geometry Points 📍

Menu

Click Points menu ➤ Create Points - Alignments ➤ Profile Geometry Points.

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreateProfileGeomPoints

Dialog Box

Create Points (page 2117)

## Creating Points on an Alignment by Importing ASCII Files

Creates points along an alignment by importing points from an ASCII (text) file that contains station, offset, and elevation information.

The file you import can contain the station, offset, elevation, and description of each point. The elevation can be expressed as either a single value (elevation) or a rod reading with instrument height (rod, hi).

ASCII (text) files that use the following layouts (formats) can be imported:

- Station, Offset
- Station, Offset, Elevation
- Station, Offset, Rod, Hi
- Station, Offset, Description
- Station, Offset, Elevation, Description
- Station, Offset, Rod, Hi, Description

Use commas or spaces as delimiters (separators) in the file. Include one or more comment lines in the file by putting a semi-colon (;) or a pound sign (#) in the first column of a comment line.

The following is an example of data in an ASCII (text) file that is formatted using the Station, Offset, Elevation format:

```
#station, offset, elevation: subdivision 1
0 20.0 112.00
10 23.5 114.64
20 22.5 116.56
30 23.0 116.32
40 22.0 115.83
```

The first line in this example is a comment line that is ignored when the points are imported. Each of the remaining lines contains the station, offset, and elevation for a point. The file is delimited by spaces.

Before you import the file, you are prompted to describe the format of the ASCII (text) file. You can also be prompted to specify invalid indicator values for station/offsets, elevations, and rod heights.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points on an alignment by importing ASCII files**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Import From File ⬡ .

3  In the Import Alignment Station And Offset File dialog box, browse to the folder containing the file. Click the file name, and click Open.

4  Enter a number between 1 and 6 to specify the format that describes the layout of data in your file, as follows:
    - 1: Station, Offset

- ■ 2: Station, Offset, Elevation

- ■ 3: Station, Offset, Rod, Hi

- ■ 4: Station, Offset, Description

- ■ 5: Station, Offset, Elevation, Description

- ■ 6: Station, Offset, Rod, Hi, Description

**5** Do one of the following:
- ■ Enter 1 if the file delimiter is a space.

- ■ Enter 2 if the file delimiter is a comma.

**6** If prompted, specify invalid indicators for station/offset, elevation, or rod/hi values.

**7** Select an alignment.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Alignments ➤ Import From File ⊕

Menu

Points menu ➤ Create Points - Alignments ➤ Import From File

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointImportFromFile

Dialog Box

Create Points (page 2117)

## Creating Points Based on Surface Elevations

Use the icons in the Surface list to access commands that you can use to create points using surface elevations.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

## Creating a Point at a Surface Elevation

Creates points at any specified location on a surface.

Point elevations are read from the specified surface.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create a point at a surface elevation**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Random Points
.

3 Select a surface.

4 Specify a location within the surface boundary for the point.
The point takes its elevation from the surface at the specified location.

5 If prompted, enter the point name and description.

6 Optionally, repeat steps 4-6.

7 Press Enter to end the command.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Random Points

Menu

Points menu ➤ Create Points - Surface ➤ Random Points

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointRandomPoints

Dialog Box

Create Points (page 2117)

## Creating a Grid of Points at Surface Elevations

Generates a grid of points that are automatically assigned surface elevations.

Specify grid base point (lower left corner), cell size (X and Y lengths), upper right corner, and optional rotation angle.
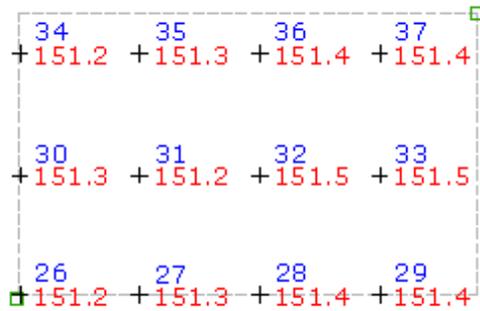
Because this command creates points using elevations from a surface, the Prompt For Elevations setting, specified in the Points Creation settings, does not affect how you are prompted during this command.

**To create a grid of points at surface elevations**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ On Grid ⊞ .

3  Select a surface.

4  Specify the location of the grid base point, which is the lower left corner of the grid, by doing one of the following:

   ■ Enter X and Y coordinates on the command line.

   ■ Specify a point on the screen.

5  Enter the rotation angle or press Enter to accept the default.

6  Specify the grid X spacing by entering a value or pressing Enter to accept the default value.

7  Specify the grid Y spacing by entering a value or pressing Enter to accept the default value.

8  Specify the location of the upper-right corner of the grid by clicking on the screen.
   A single grid square and the outline of the entire grid are drawn on the screen.

9  Do one of the following:

   ■ To create the points based on the displayed grid, press Enter.

   ■ To change the grid, enter Yes and re-enter the rotation angle, grid base point, grid X spacing, grid Y spacing, and the upper right corner as previously described.

   The command calculates the coordinates of each grid point. The elevation of each calculated point is taken from the selected surface.

10 For each point, if prompted, enter the point name and description.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ On Grid 

Menu

Points ➤ Create Points ➤ Surface ➤ On Grid

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointOnGrid

Dialog Box

Create Points (page 2117)

## Creating Points Along a Polyline or Surface Contour Using Distance and Surface Elevations

Creates points on a surface at specified distances along a polyline object or a surface contour.

Create points using elevations from the surface. The points are created along a polyline based on a specified distance.



The Prompt For Elevations setting, specified in the Points Creation settings, does not affect how you are prompted during this command.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points along a polyline or surface contour using distance and surface elevations**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Along Polyline/Contour  .

3 Select a surface.

4 Enter the distance between the points.

**5** Select the polyline or surface contour along which to place the points.

The points are placed along the polyline or surface contour at the specified interval.

**6** If prompted, for each point, enter the point name and description.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Along Polyline/Contour 

Menu

Points menu ➤ Create Points - Surface ➤ Along Polyline/Contour

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtAlongPolyContour

Dialog Box

Create Points (page 2117)

---

## Creating Points at Polyline or Surface Contour Vertices Using Surface Elevations

Creates points on a surface at the vertices of polylines or surface contours.

Points are not placed at polyline vertices that fall outside the surface boundary.

Because this command creates points using elevations from a surface, the Prompt For Elevations setting, specified in the Points Creation settings, does not affect how you are prompted during this command.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at polyline or surface contour vertices using surface elevations**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Polyline/Contour Vertices .

**3** Select a surface.

**4** Select a polyline or surface contour.

The points are placed at the polyline or surface contour vertices.

**5** If prompted, for each point, enter the point name and description.

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Surface ➤ Polyline/Contour Vertices 🏞️

Menu

Points ➤ Create Points ➤ Surface ➤ Polyline/Contour Vertices

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtPolyContourVert

Dialog Box

Create Points (page 2117)

# Creating Points by Interpolation

Use the icons in the Interpolate list to access commands that you can use to create interpolated points between selected points.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427)

## Creating Points by Interpolation

Interpolates a specified number of points between two existing points.

Interpolation determines point location and elevation. To use this command, you must have at least two points with elevations in the drawing.
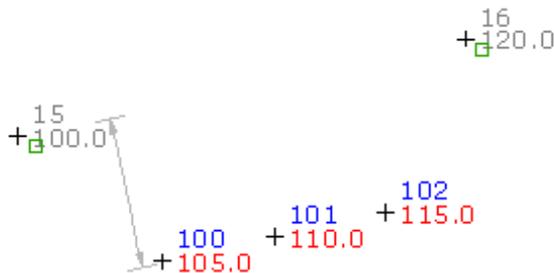


**To create a specified number of interpolated points between two points**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Interpolate ⚙.
.

**3** Specify the first point by doing one of the following:
   ■ Pick a point in the drawing.

   ■ Use Transparent commands to specify the point.

**4** Specify the second point by doing one of the following:
   ■ Pick a point in the drawing.

   ■ Use Transparent commands to specify the point.

**5** Enter the number of points to create.

**6** Enter a description, or press Enter to skip the description.


## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Interpolate ⚙

Menu

Points menu ➤ Create Points - Interpolate ➤ Interpolate

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointInterpolate

Dialog Box

Create Points (page 2117)


## Creating Points by Relative Location

Interpolates a point between two control points based on a distance.

Select an object (1) with two control points (2) and (3) and then specify a distance (4) and offset (5) to create the point.
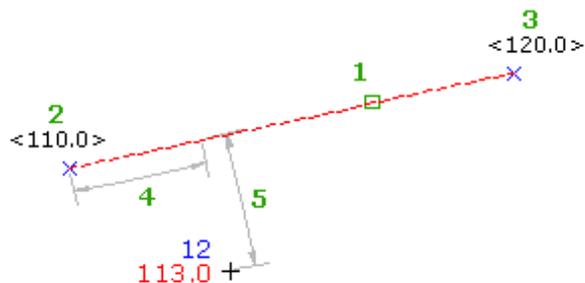
**To create points by relative location**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Interpolate ➤ By Relative Location ⬚ .

3  Specify the interpolation region by doing one of the following:

   ■  Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■  Use Transparent commands to specify the point.

   ■  Enter E, and select an arc, line, feature line, lot line, or polyline. Specify the distance and the offset distance by either entering the distance, picking a point, or using Transparent commands.

4  Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ By Relative Location ⬚

Menu

Points menu ➤ Create Points - Interpolate ➤ By Relative Location

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointByRelLocation

Dialog Box

Create Points (page 2117)

## Creating Points By Relative Elevation

Interpolates a point at a specified elevation between two control points.

Define the interpolation region by specifying the start and end control points, or click near the start point of an object.
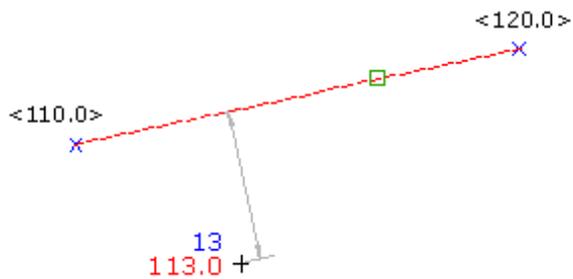
Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at an elevation by interpolating between two points**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ By Relative Elevation  .

3   Specify the interpolation region by doing one of the following:

   ■   Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■   Use Transparent commands to specify the point.

   ■   Enter E, and select an arc, line, feature line, lot line, or polyline. Enter the elevation and then specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4   Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ By Relative Elevation

Menu

Points menu ➤ Create Points - Interpolate ➤ By Relative Elevation

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointByRelElev

Dialog Box

Create Points (page 2117)

# Creating Points using Number By Distance

Interpolates a specified number of points between two control points.

Define the interpolation region by specifying the start and end control points, or click near the start point of an object.
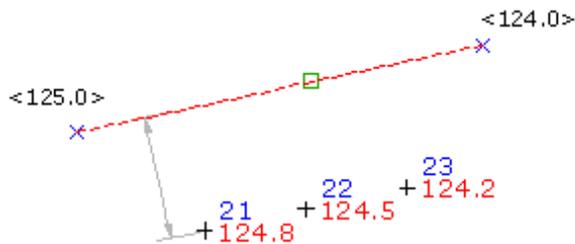


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To place a number of interpolated points**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Number By Distance � .

3   Specify the interpolation region by doing one of the following:
   ■   Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■   Use Transparent commands to specify the point.

   ■   Enter E, and select an arc, line, feature line, lot line, or polyline. Enter the number of points and then specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4   Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Number By Distance
鉤

Menu

Points menu ➤ Create Points - Interpolate ➤ Number By Distance

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

  CreatePointNumByDist

Dialog Box

  Create Points (page 2117)

## Creating Points Perpendicular

Interpolates a point along a control line or entity that is perpendicular to a specified point.

Create an interpolated point that is perpendicular along a line by selecting an object (1) that has two control points (2) and (3).
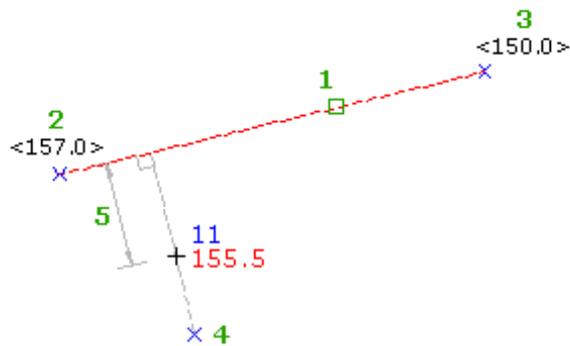


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points perpendicular from selected points**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Perpendicular
   .

3  Specify the interpolation region by doing one of the following:

   ■  Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■  Use Transparent commands to specify the point.

   ■  Enter E, and select an arc, line, feature line, lot line, or polyline. Specify the location for the perpendicular point, and then specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4  Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Perpendicular

Menu

Points menu ➤ Create Points - Interpolate ➤ Perpendicular

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtInterpPerpendic

Dialog Box

Create Points (page 2117)

## Creating Points at an Incremental Distance

Interpolates points that are a specified distance apart.

Define the interpolation region by specifying the start and end control points, or click near the start point of an object.

**To create points at an incremental distance**

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Incremental Distance 🗗 .

3   Specify the interpolation region by doing one of the following:

   ■   Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■   Use Transparent commands to specify the point.

   ■   Enter E, and select an arc, line, feature line, lot line, or polyline. Specify the distance between points, and then specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4   Enter a description, or press Enter to skip the description.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Incremental Distance 🗗

Menu

Points menu ➤ Create Points - Interpolate ➤ Incremental Distance

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtIncrementalDist

Dialog Box

Create Points (page 2117)

# Creating Points at an Incremental Elevation

Interpolates points at a specified elevation increment along a control line or entity.

Define the interpolation region by specifying the start and end control points, or click near the start point of an object.
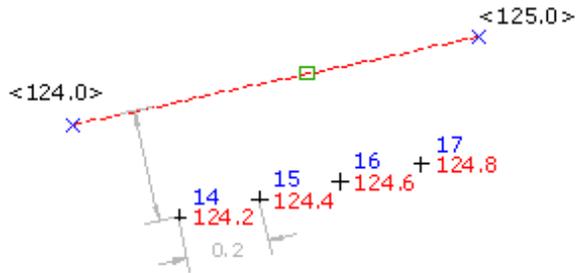


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at an incremental elevation**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Incremental Elevation ⚘ .

3  Specify the interpolation region by doing one of the following:

■  Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

■  Use Transparent commands to specify the point.

■  Enter E, and select an arc, line, feature line, lot line, or polyline. Specify the elevation difference between points, and then specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4  Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Incremental Elevation ⚘

Menu

Points menu ➤ Create Points - Interpolate ➤ Incremental Elevation

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtIncrmntlElev

Dialog Box

Create Points (page 2117)

## Creating Points at Intersections

Interpolates points where two entities intersect, or would intersect if extended.

Define the interpolation region by specifying the start and end control points, or click near the start point of an object.
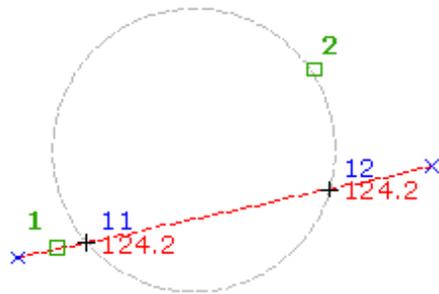


Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at object intersections**

1  Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Intersection 🔆 .

3  Specify the interpolation region for the first intersection by doing one of the following:

   ■ Pick a point in the drawing and enter an elevation for the first point. Specify the second point and then either enter D to specify the difference in elevation between the two points, or enter S to specify the slope from the first point to the second point.

   ■ Use Transparent commands to specify the point.

   ■ Enter E, and select an arc, line, feature line, lot line, or polyline. Specify the offset distance by either entering the distance, picking a point, or using Transparent commands.

4  Repeat step 4 to specify the second interpolation region.

5  In the drawing, an X marks each intersection point. If only one intersection is located then the point is automatically created.

**6** Specify the intersection point by doing one of the following:

■ Click near the X on which you want to create the point.

■ Enter A to place points at both intersections.

**7** Enter a description, or press Enter to skip the description.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Interpolate ➤ Intersection ⚹

Menu

Points menu ➤ Create Points - Interpolate ➤ Intersection

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePtInterpolIntersec

Dialog Box

Create Points (page 2117)

## Creating Points Based on Slope

Use the icons in the Slope list to access commands that you can use to create points based on slope and grade intersections, elevations, and distances.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

## Creating Points at Grade or Slope Intersections

Creates a point where two slopes or grades intersect.

Select a start and end point; specify the grade or slope ahead from the start point and back from the end point.

Creates a high or a low point between two existing point objects.

**To create a point where two slopes or grades intersect**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ High/Low Point 🔼 .

**3** Specify the start point by doing one of the following:

■ Pick a point in the drawing.

■ Use Transparent commands to specify the point.

**4** Specify the second point by doing one of the following:

- Pick a point in the drawing.

- Use Transparent commands to specify the point.

**5** Specify the slope from the first point to the second point by doing one of the following:

- Enter a slope value.

- Enter G, and then enter a grade value, or enter S and then enter a slope value.

An X displays to indicate the location of the calculated slope/grade intersection.

**6** Enter Yes to accept the calculated slope/grade intersection.

**7** Enter a description, or press Enter to skip the description.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ High/Low Point

Menu

Click Points menu ➤ Create Points - Slope ➤ High/Low Point.

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointHighLowPoint

Dialog Box

Create Points (page 2117)

## Creating Points by Slope/Grade - Distance

Creates a specified number of points from an existing point object based on the slope or grade to a given direction and distance.

Specify start point, elevation, direction, slope or grade, distance, and number of intermediate points. Offset and end point are optional.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at a given slope or grade for a specified distance**

**1** Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

**2** Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ Slope/Grade - Distance .

**3** Specify the start point by doing one of the following:

- Pick a point in the drawing.

■ Use Transparent commands to specify the point.

**4** Specify the direction and distance by doing one of the following:

■ Pick a point in the drawing.

■ Use Transparent commands to specify the point.

**5** Specify the slope from the first point by doing one of the following:

■ Enter a slope value.

■ Enter G, and then enter a grade value, or enter S and then enter a slope value.

**6** Specify the distance along the slope/grade by doing one of the following:

■ Pick a point in the drawing.

■ Use Transparent commands to specify the point.

**7** Specify the number of intermediate points from the start along the specified direction and distance.

**8** Enter Yes to add the ending point.

**9** Enter a description, or press Enter to skip the description.

---

**NOTE** Many factors affect how the point is displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

---

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ Slope/Grade - Distance 

Menu

Click Points menu ➤ Create Points - Slope ➤ Slope/Grade - Distance

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointSlopeGradeDist

Dialog Box

Create Points (page 2117)

## Creating Points by Slope/Grade - Elevation

Creates a specified number of points from an existing point object based on the slope or grade to a given direction and elevation.

Specify start point, elevation, direction, slope or grade, target elevation, and number of intermediate points. Offset and end point are optional.

For example, you can specify a grade and an ending elevation, and then create points along the distance it takes to achieve the final elevation.

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see Before You Create Points (page 427).

**To create points at a given slope or grade based on an ending elevation**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2 Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ Slope/Grade - Elevation .

3 Specify the start point by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

4 Specify the direction and distance by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

5 Specify the slope from the first point by doing one of the following:
   ■ Enter a slope value.
   ■ Enter G, and then enter a grade value, or enter S and then enter a slope value.

6 Specify the elevation along the slope/grade by doing one of the following:
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify the point.

7 Specify the number of intermediate points from the start along the specified direction and distance.

8 Specify the offset distance by doing one of the following:
   ■ Enter a distance.
   ■ Pick a point in the drawing.
   ■ Use Transparent commands to specify a point to define the offset.

9 Enter Yes to add the ending point.

10 Enter a description, or press Enter to skip the description.

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Points - Slope ➤ Slope/Grade - Elevation 

Menu

Click Points menu ➤ Create Points - Slope ➤ Slope/Grade - Elevation.

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Create

Command Line

CreatePointSlopeGradeElev

Dialog Box

[Create Points](#) (page 2117)

## Creating Points by Importing Point Data

Creates points by importing point data from a file.

Import point data from most ASCII (text) files or Microsoft® Access database *(.mdb)* files. For more information, see [Importing Point Data](#) (page 503).

## Creating Points Using the Civil Transparent Commands

Use the Civil transparent commands to create points.

You use the Civil transparent commands with the Create Points Manual command, which is included in the Miscellaneous list in the Create Points dialog box.

Create the points using known information, such as angle and distance, bearing and distance, azimuth and distance, and deflection angle and distance.

For more information about Civil transparent commands, see [Transparent Commands](#) (page 1581).

Point settings, styles, layers, point groups, and description keys can all affect how a point is created or how it is displayed in the drawing. For more information, see [Before You Create Points](#) (page 427).

**To create points using the Civil transparent commands**

1 Select settings and create styles, layers, point groups, and description keys. For more information, see [Before You Create Points](#) (page 427).

2 Click Home tab ➤ Create Ground Data panel ➤ Points Menu ➤ Create Points - Miscellaneous ➤ Manual ⊕ .

3 Do one of the following:
   ■ Enter a transparent command at the command line.
   ■ Select a transparent command from the Transparent Command toolbar.

   For more information, see [Transparent Commands](#) (page 1581).

4 Follow the transparent command prompts to place the point.

5 If prompted, enter the point name, description, and elevation.

6 Do one of the following:
   ■ Continue to enter points within the transparent command.
   ■ Press Esc to end the transparent command and return to the Create Points Manual command.

**NOTE** Many factors affect how the point is displayed. For more information, see [Controlling the Appearance of Points in a Drawing](#) (page 425).

# Editing Points

You can edit individual drawing points graphically (in the drawing), or you can use the Point Editor.

Change the properties of multiple drawing points using utilities that perform batch operations, such as renumbering or changing elevation.

You cannot edit project points directly. To edit project points, you must check the points out to a drawing, modify them, and then check them back in to the project. For a procedure for changing large numbers of project points, see Modifying A Large Number of Project Points (page 494).

For more information about working with project and drawing points, see Managing Points (page 423).

## Editing Points in a Drawing

Use AutoCAD commands to move, rotate, copy, or erase a point in a drawing.

For a complete list of the AutoCAD commands that are supported for editing points and other AutoCAD Civil objects, see AutoCAD Commands and AutoCAD Civil Objects (page 1729).

When a drawing point is locked, you cannot use any command that modifies its XYZ values, including the AutoCAD Erase command.

## Rotating Point Labels Using Grips

Use grips to rotate point labels in a drawing.

In the drawing, select a point label and click 🟢 to rotate the point label.

## Editing Points Using the Point Editor

Use the Point Editor to modify the properties of drawing points.

Each row in the Point Editor Vista displays the properties for a single point. To edit, click a cell and enter new values.

For more information about working with a vista, including keeping the window active, but hidden when you are not using it, see The Panorama Window (page 102).

The Point Editor displays point information arranged in a grid. Display and hide the columns in the Point Editor the same way you display and hide the columns in a list view. For more information, see Customizing a List View (page 84).

For a complete description of each column in the Point Editor, see Point Editor (page 2130).

Edit drawing points either in the Point Editor or from the Points list view in Prospector. The Point Editor, because it is a separate window, provides more viewing area and greater flexibility in use of screen space.

To view project points in the project Points collection, use the Prospector list view. You cannot edit project points using the list view. To edit project points, you must check the points out to the drawing, modify them, and then check them back in to the project. For more information, see Managing Points (page 423).

### Listing Points in the Point Editor

Use the Prospector shortcut menus or the List Points command to open the Point Editor.

The points that are displayed when you open the Point Editor are determined by the item or items that are selected before you display the shortcut menu. You can edit either all the points in a drawing, all the points in a point group, or selected points in a list view or drawing.

Each row in the Point Editor Vista displays the properties for a single point. To edit, click a cell and enter new values.

**To open the Point Editor**

1  In Toolspace, on the Prospector tab, do one of the following:

- To include all the points in a drawing in the Point Editor, right-click the Points collection ➤ Edit Points or click Points menu ➤ List Points.

- To include all the points in a point group in the Point Editor, right-click the point group.

- To include selected points, click the Points collection. Select the points in the list view and right-click.

- To include selected points in a drawing, select the points and right-click.

2  Click Edit Points on the shortcut menu.

The Point Editor is opened. If you cannot see the Point Editor, click ▦ at the top of the Prospector tab to display the Panorama window.

3  Use the Point Editor (page 2130) to edit or view the points.

## Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Edit/List Points ✎

Menu

Click Points menu ➤ Edit Points ➤ Points.

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Edit Points
OR
Prospector tab: Points collection ➤ right-click points in List View ➤ Edit Points

Object Shortcut menu

Edit Points

Dialog Box

Point Editor (page 2130)

## The Point Editor Shortcut Menu

Use a shortcut menu to access commands from within the Point Editor.

To display the shortcut menu, select at least one point and right-click. The commands that are available on the shortcut menu depend on whether you selected drawing or local copies of project points in the drawing, and on the state of the selected points with respect to the drawing or project.

Use the Point Editor shortcut menu to:

- Renumber selected points. (page 493)

- Change the elevation of the selected points. (page 492)

- Changing the elevations of points based on a surface. (page 493)

- Delete the selected points. (page 490)
- Zoom or pan the drawing to the selected points. (page 426)
- Lock or unlock the selected points. (page 423)

For information about the access control items on the menu, see Working with Vault Project Points (page 166).

## Deleting Points Using the Point Editor

Use the Point Editor shortcut menu to delete drawing points from within the Point Editor.

You cannot delete a locked point.

**NOTE** You can also use the AutoCAD ERASE command to delete a point.

If a deleted point is referenced by a surface, the surface will be either automatically rebuilt or marked as out-of-date, depending on the surfaceRebuild - Automatic setting.

**To delete points using the Point Editor**

1  Open the Point Editor so that it lists the point you want to delete. For more information, see Listing Points in the Point Editor (page 488).

2  Right-click the row containing the desired point. Click Delete.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Delete

Object Shortcut Menu

Erase

## Editing Points Using the Object Properties Window

Use the AutoCAD properties window to change individual COGO point properties such as layer, point style, and point label style.

**To change individual point properties**

1  Select a point in the drawing, right-click ➤ Properties.

2  On the Design tab expand the AutoCAD Civil section to display the point properties for Display, Information, Coordinate, and Labeling.

3  To change a property, click in the cell and enter a new value or select a value from the list.

## AutoCAD Civil Point Properties in the Object Properties Manager

The following AutoCAD Civil point properties are displayed in the AutoCAD Properties dialog box:

**Display**

| | |
|---|---|
| Point Style | Modify the style for the point object. If the point object does not have an assigned style <default> is displayed, indicating the point group label style is being used to draw the point label. |
| XY Scale | Modify the XY scale factor for the point object. PROPERTY NOT SET indicates this property is set to a null value. |
| Z Scale | Modify the Z scale factor for the point object. PROPERTY NOT SET indicates this property is set to a null value. |
| Rotation | Modify the point object rotation.PROPERTY NOT SET indicates this property is set to a null value. |
| **Information** | |
| Point Number | Modify the point number for the point object. |
| Name | Modify the point name for the point object. |
| Raw Description | Modify the raw description for the point object. PROPERTY NOT SET indicates this property is set to a null value. |
| Full Description | Displays the full description for the point object which is derived from the Description Format Property. This is a read only value. |
| Description Format | Modify the description format for the point object. PROPERTY NOT SET indicates this property is set to a null value. |
| Survey Point | Displays the read only value of the survey flag for the point object. A True value indicates the COGO point was created using AutoCAD Civil Survey features. |
| **Coordinate** | |
| Easting | Modify the Easting value for the point object. |
| Northing | Modify the Northing value for the point object. |
| Point Elevation | Modify the Easting value for the point object. |

**Labeling**

| | |
|---|---|
| Point Label Style | Modify the style for the point object label style. If the point object does not have an assigned style <default> is displayed, indicating the point group label style is being used to draw the point label. |
| Label Is Pinned | Specify True to pin the point label. |
| Visibility | Specify True to have the point object label displayed. |

# Editing Points Using Point Editing Utilities

Use the point editing utilities to change multiple drawing points using a single command.

You cannot use the point editing utilities to edit project points directly. To edit project points, you must check the points out to a drawing, modify them, and then check them back in to the project. For information about a recommended procedure for checking out and changing a large number of project points, see Modifying A Large Number of Project Points (page 494).

## Changing the Elevations of Points

Use the Datum command to change the elevations of selected drawing points.

Updates points that were set, according to an assumed elevation, to a benchmark elevation. If a point has no elevation value, its elevation is not adjusted.

**NOTE** Use the Point Editor to edit point elevations individually.

**To change the elevation of points**

1 In Toolspace, on the Prospector tab, click the Points collection to display the Points list view.

2 In the list view, select the desired points. For more information, see Selecting Items in a List View (page 84).

3 Right-click and click Datum.

4 In response to the prompt, specify a change in elevation by doing one of the following:

- Enter a change in elevation. To specify a decrease in the elevation, use a minus sign (-) before the value.

- Enter R, for reference, then enter the old and new datum elevations. The points will be modified by the differences between the two values.

### Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Datum

Menu

Click Points menu ➤ Edit Points ➤ Datum

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Datum

## Changing the Elevations of Points Based on a Surface

Change the elevations of a point, point group, or a selection of drawing points based on the elevations in a selected surface.

The elevations of the selected points is based on their location on the surface.

**To change the elevation of points based on a surface**

1  In Toolspace, on the Prospector tab, click the Points collection to display the Points list view, or right-click a Points collection and select Edit Points to display the Point Editor.

2  Do one of the following to display the Select Surface dialog box:
   ■ Select the points to edit in the list view or the Point Editor and right-click ➤ Elevations From Surface.
   or

   ■ Click COGO Point tab ➤ Modify panel ➤ Elevations From Surface  .

3  Select a surface from the list and click OK. If you selected the command from the list view or the Point Editor, the Point Elevation column reflects the elevation from the surface. If you selected the command from the Points menu, you are prompted at the command line to select one of the following:
   ■ All: to select all points in the current drawing to move elevations to the selected surface.

   ■ Numbers: prompts the user to specify a range of numbers.

   ■ Group: displays the Point Groups dialog box where you can select a point group.

   ■ Selection: prompts the user to select points in the drawing.

### Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Elevations From Surface 

Menu

Click Points menu ➤ Edit Points ➤ Elevations From Surface

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Elevations From Surface

## Renumbering Points

Use the Renumber command to assign new point numbers to selected drawing points.

This command adds an integer value to the point number of each selected point.

You may be prompted to resolve any numbering conflicts that arise when the renumbering occurs.

To renumber project points, check the points out of the project into a drawing, renumber them using the Renumber command, and then check them back in to the project.

**To renumber points**

1  In Toolspace, on the Prospector tab, click the Points collection to display the Points list view.

2  In the list view, select the desired points. For more information, see Selecting Items in a List View (page 84).

3  Right-click and click Renumber.

4  Enter the value to be added to the point number of the selected points.

## Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Renumber 

Menu

Click Points menu ➤ Edit Points ➤ Renumber

Toolspace Shortcut Menu

Prospector tab: Points collection ➤ right-click points in List View ➤ Renumber

Command Line

SelectAndEditPointNumbers

# Modifying A Large Number of Project Points

Use the procedure described in this topic to make a change, such as an elevation adjustment or a change to all eastings or northings, to a large number of points in a project.

To change one or more project points, you must check the points out to a drawing, change the points, and check the points back in to the project. When you are working with a large number of project points and do not need to view the points in the drawing while you change them, you can speed up the process by not drawing the point symbols or point labels while the points are in the drawing.

Avoid drawing point labels and point symbols using point label styles and a point group override. The procedure described in this topic suggests using a throw-away drawing for this process, and instructs you to edit the Standard point style and the _All Points point group in the throw-away drawing. If you do not want to use a throw-away drawing, you can modify the procedure to create a point group and a point style you can use for working with large numbers of points.

Before beginning this procedure, you should understand how to work with project and drawing points. For more information, see Working with Vault Project Points (page 166).

**To modify a large number of project points**

1  Create a new, empty drawing.

2  In Toolspace, on the Settings tab, under the Point Styles collection, right-click the Standard ➤ Edit.

3  On the Display tab, to turn off the display of point symbols, click in the Visible column for Marker so that the light bulb icon is not available.

**4** To turn off the display of point labels, click in the Visible column for Label so that the light bulb icon is not available.

**5** Click OK.

**6** In Toolspace, on the Prospector tab, under the Point Groups collection, right-click the _All Points ➤ Properties.

**7** In the Point Group Properties dialog box, on the Overrides tab (page 2113), in the Overrides column, select the Standard point style. In the Properties column, select the check box next to Point Style.

**8** In the Overrides column, select the Standard point label style. In the Properties column, select the check box next to Point Label Style.

**9** Click OK.

**10** If the drawing has more than one point group, move the _All Points point group to the top of the point group display order. For more information, see Changing the Point Group Display Order (page 526).

**11** Check out the project points to the drawing.

> **NOTE** To quickly check out all the project points, right-click the _All Points point group in the drawing and select Check Out Points.

**12** In the drawing, edit the points using either a point utility, the Point Editor, or the Prospector Points list view.

> **NOTE** For information about changing a point property such as elevation to the same value for every point in the Point Editor or the Points list view, see Changing the Contents of a Column in a List View (page 85).

**13** Check the points in to the project. Do not keep them checked out to the drawing.

> **NOTE** To quickly select all the points in the drawing for checking in, right-click the _All Points point group and select Check In Points.

**14** Delete the throw-away drawing.

# Importing and Exporting Points

You can use commands to import point data, export point data, and transfer point data between files.

Using AutoCAD Civil, you can:

■ Import points into a drawing from either an ASCII (text) file or a Microsoft® Access database (.*mdb*) file.

■ Export points from a drawing to either an ASCII (text) file or a Microsoft® Access database file.

■ Transfer points from an ASCII (text) file or a Microsoft® Access database file to another file. You can convert the point data during the transfer, which can include changing the coordinate zone.

Importing points is a quick way to place points into a drawing. For example, if a surveyor collects point data using a data collector, the data can be downloaded from the collector as an ASCII (text) file and then imported into an AutoCAD Civil drawing.

Before you can import, export, or transfer point data, you must specify a *point file format* for each file that point data is read from or written to. The point file format describes how the point data is stored in the file. You can use the point file formats that are supplied with AutoCAD Civil or create your own formats.

# Understanding Point File Formats

Use a point file format to describe how point data is arranged in a file when you import, transfer, or export points.

A *point file format* does not contain point data; it describes the layout of a *point data file,* which is a file that contains point data.

### Using Point File Formats

When you import, export, or transfer point data, you must specify a point file format for each point data file that is read or written to. When you import points from a point data file, you specify a point file format that describes how the point data is arranged in the file that is being imported. When you export point data to a file, you specify a point file format that describes how the point data is written to the exported file. When you transfer point data from one file to another, you specify two formats, one that describes how the data is arranged in the source file and one that describes how the data is written to the destination file.

### Converting Point Data During Import or Export

A point file format can contain information that specifies the coordinate zone that was used to create the point data in the associated point data file. Use the coordinate zone information to convert point data as it is imported, exported, or transferred.

A point file format can indicate that the associated point data file contains values that can be used to adjust the elevations contained within the file. The specified values can be added to or subtracted from point elevations when you import point data.

### Point File Format Types

There are two types of point file formats:

■ User Point File Format. This point file format type describes how the point data is arranged in an ASCII (text) file.

■ User Point Database Format. This point file format type describes how point data is arranged in a Microsoft® Access database file.

Use the point file formats that are supplied with AutoCAD Civil or create your own formats. The point file formats available in a drawing are listed in the Toolspace Settings tree.

# Point File Formats Collection (Settings Tab)

Use the Point File Formats collection in the Settings tree to create and manage point file formats.

In the Settings tree, expand the Point collection and then expand the Point File Formats collection to list the point file formats available in the drawing.

Right-click the Point File Formats collection to:

■ Create a new point file format (page 500).

■ Refresh the Settings tree.

Right-click a point file format name to:

■ Change the properties of the point file format. (page 497)

■ Make a copy of the point file format. (page 502)

■ Delete the point file format. (page 503)

- Refresh the Settings tree.

You can also list available point file formats by clicking the Point File Formats collection to display a list view. For more information, see The Toolspace Item View (page 82).

## Point File Format Properties

Use a dialog box to view or change the properties of a point file format.

The properties of a point file format include:

- format file name

- default point data file extension

- comment tag

- transformation information

- column names that describe the layout of the point data in the associated point data file

- file formatting options (point file format only)

The dialog box you use to view and change the properties of a point file format depends on whether the format is a user point file format or a user point database format. For more information about the dialog box used to view and change the properties of a point file format, see Point File Format Dialog Box (page 2120). For more information about the dialog box used to view and change the properties of a user point database format, see User Point Database Format Dialog Box (page 2121).

For information about creating a new point file format, see Creating Point File Formats (page 500).

## Point File Format Coordinate Zone Transformation Property

Use the coordinate zone transformation property in a point file format to specify the coordinate zone in which the data in the associated point data file was created.

When you import or export a point data file, coordinate zone information is included in the point file format, not the point data file. Use this information to transform points from one coordinate zone to another while importing or exporting.

For example, to import an ASCII (text) file that was created in an NAD27 zone when the zone of the current drawing is NAD83, specify the NAD27 zone in the point file format. Then, when you import the points, you can select an option that converts the points to the drawing zone, NAD83.

## Point File Format Column Name Property

Use column names in a point file format to specify which point information is included in a point data file.

When creating a new point file format, you are presented with a list of unused columns:



Specify the contents of the first column in the point data file by clicking the first column heading and specifying a point property, for example, Easting, in the Point File Formats - Select Column Name dialog box. The first column heading is updated to reflect the specified column name:

Specify the contents of the second column in the point data file by clicking the second column heading in the dialog box and specifying another point property, for example, Northing:





Continue to specify column headings until the point file format column names reflect the layout of the point data file.

Some point file format columns can be used to perform calculations when they are included in the format. For more information, see Using Point File Format Properties to Perform Calculations (page 499).

## Point File Formatting Options

In a point file format, use formatting options to specify whether the associated point data file is arranged in columns or delimited by a specified character.

Choose either Columnated or Delimited By to specify how point data is stored in a point data file.

### Columnated

Use the Columnated option to separate values in the point data file by tabs. When you import the file, the data in the file is assumed to be arranged in columns and rows. Each row contains the data for a single point, and the individual values in the row are separated by tabs. When you export points or transfer data, the values in the resulting file are arranged in columns and separated by tabs, with one point per row.

For example, if your format column names are Number, Northing, Easting, and Elevation, the contents of your point data file might appear as follows:

| | | | |
|------|---------|---------|--------|
| 1002 | 5000.00 | 2010.00 | 100.00 |
| 1003 | 5001.00 | 2020.00 | 100.00 |
| 1004 | 5002.00 | 2030.00 | 100.00 |

### Delimited

The Delimited By option separates the values in the point data file by a delimiter, such as a comma or a space. When you import the file, each row in the file is assumed to contain data for a single point, with the individual values separated by the specified delimiter. When you export points or transfer data, the values in the resulting file are written to the file, one point per row, with the individual values separated by the specified delimiter.

For example, if your format column names are Number, Raw Description, Northing, Easting, and Elevation, and you specified a comma (,) for a delimiter, the contents of your point data file might appear as follows:

101,MONA,300,500,0

102,MONB,302,499,0

103,MONA,303,501,0

# Using Point File Format Properties to Perform Calculations

Use some columns in a point file format to perform calculations, such as adjusting the elevation values.

## Adjusting Elevation During Import and Transfer

During point import, use some format column names to adjust point elevation values during point import.

Use the following columns:

■ Thickness: Include this column name in the format if your data includes surface thickness values, such as the thickness of a layer of topsoil or clay. The value in the Thickness column can be subtracted from the value in the Elevation column when points are imported or transferred.

■ Z+: Include this column name in the format to define a column for point data that was taken at a height above the datum elevation. The value in the Z+ column can be added to the value in the Elevation column when the points are imported or transferred.

■ Z-: Include this column name to adjust the elevation of a point in a file. The value in the Z- column for a point can be subtracted from the value in the Elevation column when the points are imported or transferred.

The values specified for Thickness, Z+, or Z- in the point data file are not automatically used to adjust the elevation during import or export. Select the Do Elevation Adjustment If Possible option when importing points when you want elevation adjustments to occur.

## Calculating Convergence Angles During Export

Include the Convergence column in a point file format that is used for exporting points.

Use the Convergence column name to define a point file format column containing a calculated convergence angle. In state plane coordinate systems, the convergence angle is the difference between a geodetic azimuth and the projection of that azimuth onto a grid (grid azimuth) of a given point.

Use this column only when you export point data. To export points using a format containing a convergence angle, you must have set a zone in the file format and drawing, and you must turn on transformation settings for the drawing.

## Calculating Scale Factors During Export

Include the Scale Factor column in a point file format that is used for exporting points.

Use the Scale Factor column name to define a point file format column containing a calculated convergence angle. Scale factor is the value used to reduce or increase a local (geodetic) distance so that it equals the grid distance.

Use this column only when you export point data. To export points using a format containing a scale factor, you must have set a zone in the file format and drawing, and you must turn on the transformation settings for the drawing.

Export the scale factor by using the method specified in the transformation settings, such as Prismoidal or Unity.

# Managing Point File Formats

Use the Settings tree to manage point file formats. You can share a point file format and print a list of the point file formats in a drawing.

## Sharing Point File Formats

Use the Settings tree to list the point file formats for a drawing.

Point file formats are associated with a drawing. If more than one person will need to use a point file format, you can create it in a drawing template. For more information, see Drawing Templates (page 109).

## Printing Point File Formats

Use the Copy To Clipboard command to copy the list of the point file formats in a drawing to another application for printing.

To print a list of the point file formats in a drawing, click the Point File Formats collection in the Settings tree to display a list view containing the point file formats in the drawing. Right-click the list view and click Copy To Clipboard to copy the list view into a file you can print. For information, see Copying Items from a List View (page 86).

# Creating Point File Formats

Create two types of point file formats: a user point file format and a user point database format.

A point file format does not contain point data; it describes the way point data is stored in a point data file. When you create a point file format, regardless of whether it is a user point file format or a user point database format, you can specify the following properties:

■ A format name, which is displayed in the Settings tree.

■ A series of column names that describe both the point information stored in the point data file and the order in which the point information is stored.

■ The coordinate zone in which the point data was created. This information is optional, but if you include it, you can automatically perform a coordinate transformation of the point data if you import it into a drawing that uses a different coordinate zone, or if you transfer the points from one file to another.

## Creating a User Point File Format

Create a user point file format for importing or exporting ASCII files that contain point data.

When you create a user point file format, you specify the information that is included in the point data file, the layout of the information in the file, and the character that is used to separate the individual pieces of information, such as a comma or a space. For more information, see Point File Format Properties (page 497).

As you create a user point file format, you can view the file that contains your point data for reference.

**NOTE** If you are importing point data from and ASCII file that contains user defined columns, you must create a user defined classification for each column. For more information, see Importing Point Data from an ASCII File Containing User Defined Columns.

**To create a user point file format**

1   In Toolspace, on the Settings tab, right-click the Point File Formats ➤ New.

2   In the Point File Formats - Select Format Type (page 2120) dialog box, click User Point File. Click OK.

3   In the Point File Format (page 2120) dialog box, specify a name and file extension for the format.

4   Click Load to load a point data file so that you can refer to it as you create the format.

**NOTE** At any time, you can click the Parse button and the loaded file is displayed in the dialog box using the format you are building.

5 Optionally, specify a comment tag character and a coordinate zone transform.

6 Specify the format options for the file.

7 Click a column name heading that is labeled <unused>.

8 In the Point File Formats - Select Column Name (page 2122) dialog box, select a column name from the Column Name list. Specify other parameters as needed, and click OK.

9 Repeat the previous step for each column in the point file format.

10 Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Point File Formats collection ➤ New

Command Line

CreatePointFormat

Dialog Box

Point File Format (page 2120)

## Creating a User Point Database Format

Create a user point database format for importing or exporting *.mdb* files that contain point data.

When you create a user point database format, you specify both the information included in the *.mdb* file and the order in which it is included. If the file has multiple tables, you specify the name of the table from which you want to import data or to which you want to export data.

As you create a point database format, you can view a sample point database for reference.

**To create a user point database format**

1 In Toolspace, on the Settings tab, right-click the Point File Formats ➤ New.

2 In the Point File Formats - Select Format Type (page 2120) dialog box, click User Point Database and click OK.

3 In the User Point Database Format (page 2121) dialog box, specify a name for the format.

4 Click Load to load the Microsoft® Access file that you want the new format to match, so that you can refer to it as you create the format.

5 Specify a table name if required.

6 Optionally, specify a coordinate zone transform.

7 Click a column name heading that is labeled <unused>.

8 In the Point File Formats - Select Column Name (page 2122) dialog box, select a column name from the Column Name list. Specify other parameters as needed, and click OK.

9 Repeat the previous step for each column in the user point database format.

**10** Click OK.

### Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click Point File Formats collection ➤ New

Command Line

  CreatePointFormat

Dialog Box

  User Point Database Format (page 2121)

## Creating a Point File Format Based on an Existing Format

Creates a point file format based on an existing format by copying the format.

After you make a copy of a format, the format you created is automatically opened for editing.

**To create a point file format based on an existing format**

**1** In Toolspace, on the Settings tab, expand the Point File Formats collection.

**2** Right-click the point file format you want to copy. Click Copy.

**3** Do one of the following:

- If the format you copied was a user point file format, use the Point File Format (page 2120) dialog box to change the format properties.

- If the format you copied was a user point database format, use the User Point Database Format (page 2121) dialog box to change the format properties.

### Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click Point File Formats item ➤ Copy

## Editing Point File Formats

Use the Settings tree to access formats you want to change or delete.

You can change or delete point file formats that you create or most of the point file formats AutoCAD Civil supplies. A point file format that cannot be changed or deleted has a 🔒 displayed to the left of its name in the Prospector tree.

## Changing a Point File Format

You can change any point file format that you create. You can also change most of the point file formats that are supplied with AutoCAD Civil.

A point file format cannot be changed if 🔒 is displayed to the left of its name in the Prospector tree.

**To change a point file format**

1  In Toolspace, on the Settings tab, expand the Point File Formats collection.

2  Right-click the desired point file format. Click Properties.

3  Do one of the following:

   ■  If the format is a point file format, use the Point File Format (page 2120) dialog box.

   ■  If the format is a user point database format, use the User Point Database Format (page 2121) dialog box.

4  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Point File Formats item  ➤  Properties

Dialog Box

Point File Format (page 2120),
User Point Database Format (page 2121)

## Deleting a Point File Format

Use the Settings tree to delete a point file format from a drawing.

If a point file format cannot be deleted, ⊕🔒 is displayed to the left of its name in the Prospector tree.

**To delete a point file format**

1  In Toolspace, on the Settings tab, expand the Point File Formats collection.

2  Right-click the desired point file format. Click Delete.

3  Click Yes to confirm.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Point File Formats item  ➤  Delete

## Importing Point Data

Import point data from an ASCII (text) file or a Microsoft® Access *.mdb* file into a drawing.

Importing points, such as survey point data collected by a data collector, is a quick and effective way to place points into a drawing.

Before you import points, you must create a point file format that describes the layout of the point data in the point data file.

For more information, see Understanding Point File Formats (page 496) and Creating Point File Formats (page 500).

You can add the imported points to a point group, make adjustments to the data as it is imported, including elevation adjustments, coordinate transformation, or coordinate data expansion, encounter how the imported points are numbered as they are created.

**To import point data**

1 Specify the Point Identity settings, which control the point numbers of the created points. For more information, see Editing the Point Identity Settings (page 413).

2 Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 .

3 On the Create Points (page 2117) toolbar, click the Import Points button.

4 In the Import Points (page 2124) dialog box, specify the point file format that describes the layout of the data in the point data file you are importing.

5 In the Import Points (page 2124) dialog box, click ➕ , and select the point data file you want to import.

**NOTE** To migrate points from Autodesk Land Desktop, select External Project Point Database as the format type and select the Autodesk Land Desktop point database .mdb file as the source file.

6 Optionally, specify a point group to which the imported points are added.

7 Optionally, specify advanced options for elevation adjustment, coordinate transformation, or coordinate data expansion.

8 Click OK to import the points.

## Quick Reference

Ribbon

Click Insert tab ➤ Import panel ➤ Points From File 🔧

Menu

Click Points menu ➤ Import/Export Points ➤ Import Points

Toolspace Shortcut Menu

Prospector tab: right-click Point collection ➤ Create ➤ Import Points

Command Line

ImportPoints

Dialog Box

Create Points (page 2117)

## Importing Point Data from an ASCII File Containing User Defined Columns

Before you import point data from an ASCII (text) file that contains user-defined columns you must create a User-Defined Property Classification.

**To import point data from and ASCII file that contains user-defined columns**

1  In Toolspace, on the Settings tab expand the Point collection ➤ right-click User-Defined Property Classification ➤ New. For more information, see Creating a User-Defined Property Classification (page 417).

2  Right-click the classification that you created in the previous step and click New to create a new user-defined property. For more information, see Creating a User-Defined Property (page 417).

3  In Toolspace, on the Settings tab expand the Points collection and click Point File Formats ➤ New.

4  Select User Point File in the Point File Formats - Select Format Type dialog box and click OK.

5  Create the new point file format. For more information, see Creating Point File Formats (page 500).

6  To specify the user-defined columns, click an <unused> column heading to display the Point File Formats - Select Column Name dialog box.

7  In the Column list, select the user-defined property you created in step two.

8  On the Prospector tab click Points ➤ Create ➤ Import Points. Select the ASCII file you wish to import and select the Point File Format you defined in step five.

# Exporting Point Data

Export drawing points to an ASCII (text) file or a Microsoft® Access *.mdb* file.

Before you export points, you must create a point file format that describes the layout of the point data in the point data file. By default, all points in the drawing are exported. To export only selected points, select Limit To Points In Point Group.

A point file format describes how the point data is arranged in the file. For example, in a drawing, point properties include point number, name, description, northing, easting, and elevation. If you want to export only point number, description, and elevation, you must export points using a point file format that defines column names only for point number, description, and elevation. For more information, see Understanding Point File Formats (page 496) and Creating Point File Formats (page 500).

In addition to exporting point properties, you can also export grid northing, grid easting, latitude, and longitude if a coordinate zone and transformation settings are defined for the drawing.

You can perform a coordinate transformation on the points as you export them. To export the point data so that it is written to the point data file using a different coordinate zone from the current drawing, specify an option that performs a coordinate transformation of the point data as it is exported. To transform the point data, your point file format must have a coordinate zone defined. For more information, see Point File Format Coordinate Zone Transformation Property (page 497).

You can export either all the points in a drawing or you can export selected drawing points using a point group.

**To export point data**

1  Create a point file format that describes the layout of the exported data file. For more information, see Creating Point File Formats (page 500).

2  Do one of the following:

■  Click Output tab ➤ Export panel ➤ Export Points ⬚ to export all the points in a drawing.

■  In Toolspace, on the Prospector tab, expand the Point Groups collection. Right-click the point group name ➤ Export Points to export only the points in a point group.

**3** In the Export Points (page 2126) dialog box, in the Format list, select the point file format that describes how you want the exported point data to be arranged in the point data file.

**4** Click  to specify the file to which you want to export the data.

**5** Optionally, specify advanced options.

**6** Click OK to export the points.

## Quick Reference

Ribbon

Click Output tab ➤ Export panel ➤ Export Points 

Menu

Click Points menu ➤ Import/Export Points ➤ Export Points

Toolspace Shortcut Menu

Prospector tab: right-click Points collection ➤ Export

Command Line

ExportPoints

Dialog Box

Export Points (page 2126)

# Transferring and Converting Point Data

Use point file formats to transfer point data from one format to another and to convert the coordinate system of a set of points.

## Transferring Point Data Between Files

Use the Transfer Points utility to transfer point data from a source file to a destination file. Either file can be an ASCII (text) file or a Microsoft® Access database file.

You must create two point file formats. The first file format describes the layout of the data in the source file, which is the file the data is read from. The second file format describes the layout of the data in the destination file, the file the data is written to.

For example, your source file point file format could indicate that the source point data file contains the following point properties in this order: point number, elevation, northing, easting, and description. Your destination file point file format could indicate that the destination file should contain the following point properties in this order: northing, easting, elevation. In this example, when you transfer the data from the source file to the destination file, the point number and description are not transferred, and the position of the elevation value is changed.

When you transfer data between two files, the points that are transferred are not deleted from the source file.

The point file format type must match the source or destination file type. For example, to transfer points from a Microsoft Access database file to an ASCII (text) file, specify a user point database format that describes the layout of the data in the Microsoft Access database (the source file) and a point file format that describes the layout of data in the ASCII file (the destination file).

**To transfer point data between files**

**1** Create point file formats for the files you want to transfer points between. For more information, see Creating Point File Formats (page 500).

**2** Click Output tab ➤ Export panel ➤ Transfer Points ⬚ .

**3** In the Transfer Points (page 2126) dialog box, under Source, specify the format and file name, including the path, for the Source file.

**4** Under Destination, specify the format and file name, including the path, for the Destination file.

**5** To create a new point file format or change an existing one, click Manage.

**6** Optionally, specify advanced options.

**7** Click OK to transfer the point data from the source file to the destination file.

## Quick Reference

Ribbon

Output tab ➤ Export panel ➤ Transfer Points ⬚

Menu

Points menu ➤ Import/Export Points ➤ Transfer Points

Command Line

TransferPoints

Dialog Box

Transfer Points (page 2126)

## Converting Points to a Different Coordinate Zone

Use a combination of the Point Import, Transfer, and Export commands to convert points to a different coordinate zone.

You convert the points by exporting them to a file, changing the current zone of the drawing, and importing the points back into the drawing and overwriting the original point data.

**To convert points to a different coordinate zone**

**1** Create a copy of the External Project Point Database format. For more information, see Creating a Point File Format Based on an Existing Format (page 502).

**2** Change the coordinate zone of the new format so that it matches the coordinate zone of the current drawing. For more information, see Changing a Point File Format (page 502).

**3** Export the points in the drawing using the new format. For more information, see Exporting Point Data (page 505).

**4** Change the current drawing coordinate zone to the zone you want the points to be converted to.

**5** Import the points into the drawing. For more information, see Importing Point Data (page 503).

# Using External Data References

AutoCAD Civil creates and maintains a point database file that contains all the point information in the project.

COGO points store the point number, name (optional), northing, easting, elevation, and description in each drawing.

You can also reference your own customized databases and use them to do the following:

■ Substitute point elevation data when points are accessed through a point group.

■ Substitute point raw description data.

To link these custom point databases to AutoCAD Civil, create External Data References (XDRefs). For more information, see Creating an External Data Reference (page 509). An XDRef is a pointer to an entire column of data in a custom Microsoft Access database. All of the database entries must have a point number. Then, when you use an XDRef to get a value for a point, the point number is looked up in the custom database, and the value from the specified column is used instead of the original point value that is stored in the drawing.

XDRefs do not overwrite or alter the points in the drawing. Use the External Data Reference collection on the Settings tree in Toolspace to create and manage XDRefs.

## External Data Reference Requirements

The custom databases that you can use for XDRefs must have the following features:

■ They must be Microsoft Access database files.

■ There must be a Long Integer field index column, which contains the point numbers, in each table in the database that can be used as an XDREF.

■ Currently, only Integer, Long Integer, Single, Double, and Text type fields are supported.

■ Any number of Tables can be defined in this database, but any that are referenced by XDRefs must have an index column defined.

■ Any number of additional Text or Number columns may also be defined in this database table. There are no restrictions on the names and order of the columns after the index column.

■ XDRefs are stored on the Settings tab in Toolspace under the Point collection.

## Creating an External Point Database with Microsoft Access

Use Microsoft Access to create custom point databases, and link these databases to AutoCAD Civil by using external data references (XDRefs).

You can modify the sample database located in *c:\program files\AutoCAD Civil\Sample\Civil 3D XDRef\* or create a new database.

**To create an external point database**

1 Start Microsoft Access.

2   Select Blank Database and click OK.

3   In the File New Database dialog box, use the Save In list to locate a folder for future use.

4   In the File Name box, enter a name for the database.

5   Click Create to display the Database dialog box.

6   On the Table tab, click New to open the New Table dialog box.

7   Select Design View.

8   Click OK to display the design view of the table.

9   In the first table cell in the Field Name column, enter a name for the Index Column.

10  In the first table cell in the Data Type column, select Number.

11  In the lower part of the dialog box, verify the following information:

    ■ Long Integer is the Field Size

    ■ Auto is the Decimal Places

12  In the Required field, select Yes.

13  In the Indexed field, select Yes (No Duplicates).

14  Add any additional field names below the Index Column as needed for your point information, such as DESC 1, DESC 2, ELEV 1, ELEV 2. There are no restrictions on the names and order of the columns after Index Column.

15  In the first table cell in the Field Name column, enter the name of the Index Column.

16  In the first table cell in the Sort Order column, select either Ascending or Descending.

17  In the lower part of the dialog box, change the following settings:

    ■ Primary field must be Yes.

    ■ Unique field must be Yes.

    ■ Ignore Nulls field must be No.

18  From the View menu, choose Datasheet.

19  Click Yes to display the Save As dialog box.

20  Enter a name for the table and click OK.

21  In the Table dialog box, enter point information such as point numbers, elevations, and descriptions.

22  Click File menu ➤ Save.

23  Close Microsoft Access.

## Creating an External Data Reference

External Data References (XDRefs) are associated with individual drawings. You can use XDRef's to override the raw description and elevation for a point.

**To create an external data reference**

1   On the Settings tab in Toolspace, expand the Points folder.

**2** Right-click External Data References ➤ New.

**3** Enter a name and description.

**4** Optionally, if the database is password protected, enter the password to access the database.

**5** Browse to the location of the external data file.

**6** Select the Table type, Index Column, and Value Column.

**7** Click OK. The new external data reference is listed in the External Data Reference folder under Points in Toolspace.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click External Data Reference collection ➤ New

Command Line

CreateXdref

Dialog Box

(page 2136)

## Changing the Properties of an External Data Reference

Edit an existing XDRef by changing its properties, such as the table and column in the Microsoft® Access database that the XDRef points to.

**To change the properties of an external data reference**

**1** On the Settings tab in Toolspace, expand the Points folder.

**2** Right-click External Data References ➤ Properties.

**3** Enter a new name and description.

**4** Select new values for the Table type, Index Column, and Value Column.

**5** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click External Data Reference collection ➤ Properties

Dialog Box

Create External Data Reference Dialog Box (page 2136)

## Point Utilities

you can use these commands to view and zoom to points in the project, to draw project extents, as well as to create BLOCKS from COGO points and to export points.

# Quick View Project

Use this command for a quick graphic view of all the point objects in the drawing.

The drawing must be connected to a project for this command to work, but you do not need to have the drawing checked out.

**To view point objects in a drawing**

■  Click Points menu ➤ Utilities ➤ Quick View Project.

## Quick Reference

Menu

  Points menu ➤ Utilities ➤ Quick View Project

Command Line

  QuickViewProjectPoints

# Draw Project Extents

Use this command to view the extents of points in the project.

The drawing must be connected to a project for this command to work, but you do not need to have the drawing checked out.

The program draws a rectangle outlining the project extents.

**To draw the project extents**

■  Click Points menu ➤ Utilities ➤ Draw Project Extents.

## Quick Reference

Menu

  Points menu ➤ Utilities ➤ Draw Project Extents

Command Line

  ShowProjectPointExtents

# Zoom to Project Extents

Use this command to zoom to the extents of the project.

The drawing must be connected to a project for this command to work, but you do not need to have the drawing checked out.

**To zoom to the project extents**

■  Click Points menu ➤ Utilities ➤ Zoom to Project Extents.

### Quick Reference

Menu

Points menu ➤ Utilities ➤ Zoom To Project Extents

Command Line

ZoomToProjectPointExtents

## Converting Softdesk Point Blocks to AutoCAD Civil Points

Convert Softdesk point blocks to AutoCAD Civil points.

COGO points are created using elevations and raw description values contained within each Softdesk point block reference. When the points are converted to COGO points, the full description and raw description are the same.

### To convert Softdesk point blocks to AutoCAD Civil points

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Replace Softdesk Point Blocks .

3   Select the Softdesk point blocks to be converted. Press Enter.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Replace Softdesk Point Blocks

Menu

Points menu ➤ Utilities ➤ Replace Softdesk Point Blocks

Command Line

CreatePtConvertSdskPts

## Converting AutoCAD Points to AutoCAD Civil Points

Convert AutoCAD point objects to AutoCAD Civil points.

Each resulting COGO point acquires its elevation from the specified AutoCAD point.

### To convert AutoCAD points to AutoCAD Civil points

1   Select settings and create styles, layers, point groups, and description keys. For more information, see Before You Create Points (page 427).

2   Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Convert AutoCAD Points .

3   Select the AutoCAD points to be converted.

4   For each point, if prompted, enter the point name and description.

**NOTE** If you do not want to be prompted for a description for each point, in the Points Creation section of the Edit Point Settings dialog box set the Prompt for Descriptions Property to None. You can also use this procedure when you want to disable Prompt for Point Names and Prompt for Elevation.

**5** Optionally, repeat steps 4 and 5.

**6** Press Enter to end the command.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Convert AutoCAD Points ⊕

Menu

Points menu ➤ Utilities ➤ Convert From AutoCAD Points

Command Line

CreatePtConvertAdeskPts

# Creating Blocks from COGO Points

Use this command to create an AutoCAD block for each AutoCAD Civil point in a selection set.

The blocks can be used in backward migration to other AutoCAD programs, or to create objects that represent the spatial location of specified civil points in the active drawing.

**To create blocks from COGO points**

**1** Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Blocks From COGO Points ⊞ .

**2** Specify a Spatial Filter.

**3** Under Selected Point Groups, click ⊕ and specify the point groups you want included in the block.

**NOTE** You can select the points to include in the block using both the Spatial Filter and Point Group options. For example, under Spatial Filter you could select the Current Display and under Point Groups you could also include a specified point group.

**4** In the Block Output section, under Block Creation, click either Use Existing Block or Make New Block. If you select Make New Block, enter a name.

**NOTE** The option Use Existing Block is only available if the block contains at least three attribute definitions with the exact names ELEV, POINT, and DESC. If there are no blocks in the drawing with these exact attribute definitions, this option is not available. Then you must use the Make New Block option.

**5** Under Block Layer, specify the layer on which you want to create the block.

**NOTE** The COGO points are not deleted when the blocks are created.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Blocks From COGO Points ⌖

Menu

Points menu ➤ Utilities ➤ Create Blocks From COGO Points

Command Line

ConvertPointsToSdskPoints

Dialog Box

Create Blocks from COGO  (page 2136)

## Converting Land Desktop Points

Use the convert Land Desktop Points command to convert AutoCAD Land Desktop points in a drawing to AutoCAD Civil points.

When you use AutoCAD Civil to open a drawing that contains Land Desktop points, you can run this utility to perform the conversion.

Before converting the points, you can use the Convert Autodesk Land Desktop Points dialog box to set a variety of point settings. For example, you can set default layers, point creation settings, and more. For more information, see Configuring Land Desktop Point Settings.

You can automatically add the converted points to a newly created point group or to an existing point group. The existing AutoCAD Civil point number and point name conflict resolution rules are used if a point ID conflict occurs during the conversion.

**To convert Land Desktop points**

1 Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Convert Land Desktop Points to display the Convert Autodesk Land Desktop Points dialog box.

2 Configure point setting parameters by expanding the parameter, selecting a setting, then specifying a new value in the Value column.

3 To assign the Land Desktop points to a point group, select Add Points To Point Group, then choose a point group from the list or create a new point group. If you create a new point group, it is added to the Add Points To Point Group list.

4 To keep the existing layers referenced by the Land Desktop points, select Preserve Original Point Layer.

> **NOTE** When this check box is checked, the existing layer of the Land Desktop point is assigned to the AutoCAD Civil point when it is converted. If the Points Creation\Disable Description Keys setting is set to False, and the Land Desktop point matches a description key, the original point layer will still be preserved.

5 Click OK.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Convert Land Desktop Points

Menu

Points menu ➤ Utilities ➤ Convert Land Desktop Points

Command Line

ConvertLDTPoints

Dialog Box

Convert Land Desktop Points dialog box (page 2137)


# Geodetic Calculator

Use the Geodetic Calculator to calculate geodetic information relative to the zone and transformation values specified in the drawing settings.

A Coordinate System zone must be specified for the drawing in order to use this command. Transformation settings must be specified in order to view/calculate local coordinates relative to the Latitude/Longitude and/or Grid coordinates.

Enter one pair of known coordinate values, either Local Northing/Local Easting, Latitude/Longitude, or Grid Northing/Grid Easting. The other two coordinate pairs are calculated.

For example, if you enter a value in the Local Northing value field, the Local Northing/Local Easting coordinate pair is assumed to contain the known data, and the values for the Latitude/Longitude and Grid Northing/Grid Easting pairs are calculated. If you enter a value into either the Latitude or Longitude value field, the Latitude/Longitude coordinate pair is assumed to contain the known data, and values for the Local Northing/Local Easting and Grid Northing/Grid Easting pairs are calculated.

If you enter a point number or select a point location in the drawing, the Local Northing/Local Easting pair is updated, and the Latitude/Longitude and Grid Northing/Grid Easting pairs are calculated.

**NOTE** You can run other commands while the Geodetic Calculator is active.

**To use the Geodetic Calculator**

1 Click Modify tab ➤ Points ➤ Geodetic Calculator.

2 In the Geodetic Calculator (page 2133) dialog box, specify starting information by doing one of the following:

■ Click ⬚ to select a point location in the drawing.

■ Enter a point number.

■ In the Value column of the Geodetic Calculator, enter a set of coordinate values, either Latitude and Longitude, Grid Northing and Grid Easting, or Local Northing and Local Easting.

3 Optionally, click ⊕ to create a point with the values for the Local coordinates you entered or calculated. You are prompted at the command line to enter a point description and elevation.

4 To perform additional calculations, enter a value for a coordinate field, such as Local Northing.

5 After you enter the value, click the Value column for another coordinate field, such as Grid Easting.

6 Update all values by clicking in another field.

**Quick Reference**

Ribbon

Click a point in the drawing Analyze tab ➤ ➤ COGO points tab ➤ Geodetic Calculator 

Menu

Survey menu ➤ Geodetic Calculator

Command Line

ShowGeodeticCalculator

Dialog Box

Geodetic Calculator (page 2133)

# Point Command Reference

You can use these commands to quickly access point-related functionality.

The following table lists the point-related AutoCAD Civil commands and briefly describes their functionality. For more information about a command, follow the link in the Description column.

| Command | Description |
| --- | --- |
| AddPointTable | Adds a point table to a drawing. (page 422) |
| AeccCreatePtGeodeticDir | Creates Points specified by geodetic directions. (page 429) |
| ConvertPointsToSdskPoints | Creates an AutoCAD block for each AutoCAD Civil point in the selection set. Use these blocks for backward migration. (page 513) |
| CreateDescKeySet | Creates a description key set. (page 547) |
| CreatePointAlignAlign | Creates a point at the intersection of two alignments. (page 458) |
| CreatePointAtPtPcScetc | Creates a point at every intersection on the alignment. (page 463) |
| CreatePointAutomatic | Creates points at the endpoints of lines, feature lines, or lot lines, or at the endpoints and center point of arcs. (page 432) |
| CreatePointByRelElev | Interpolates point objects between two points based on desired elevations. (page 476) |
| CreatePointByRelLocation | Interpolates point objects between two points based on distance from the first location. (page 475) |
| CreatePointDirectionAlign | Creates a point at the intersection of a direction line and an alignment. (page 453) |

| Command | Description |
|---------|-------------|
| CreatePointDirectionDir | Creates a point at the intersection of two lines. (page 440) |
| CreatePointDirectionPerp | Creates a point perpendicular on a line definition. (page 444) |
| CreatePointDirectionObject | Creates a point at the intersection of a line and an object. (page 447) |
| CreatePointDistanceAlign | Creates a point at the intersection of a circle and an alignment. (page 455) |
| CreatePointDistanceDist | Creates a point at the intersection of two circles. (page 441) |
| CreatePointDivideAlign | Create a specified number of evenly spaced points relative to an alignment. (page 461) |
| CreatePointDivideObject | Creates a specified number of evenly spaced points along an object. (page 435) |
| CreatePointFormat | Creates a point file format. (page 500) |
| CreatePointGroup | Creates a point group. (page 529) |
| CreatePointHighLowPoint | Adds a high or a low point between two existing point objects. (page 483) |
| CreatePointImportFromFile | Creates points on an alignment using point data in an ASCII (text) file. (page 468) |
| CreatePointInterpolate | Interpolates two points with known elevations. (page 474) |
| CreatePointManual | Creates a point at a specified location in the drawing. (page 428) |
| CreatePointMeasureAlig | Creates points at equal intervals along an alignment. (page 462) |
| CreatePointMeasureObject | Creates points along an object using a specified distance. (page 437) |
| CreatePointNumByDist | Interpolates a specified number of point objects between two points. (page 478) |
| CreatePointObjectAlign | Creates a point at the intersection of an object and an alignment. (page 456) |
| CreatePointObjectObject | Creates a point at the intersection of two objects. (page 450) |
| CreatePointOnGrid | Creates points in a grid configuration using elevations from a surface. (page 484) |

| Command | Description |
| --- | --- |
| CreatePointRadialOrPerp | Creates points radial or perpendicular to an alignment. (page 467) |
| CreatePointRandomPoints | Creates a point at a specified location using an elevation from a surface. (page 483) |
| CreatePointResection | Creates a point at a position that is calculated from the measured angles between three known points. (page 430) |
| CreatePointSlopeGradeDist | Creates a specified number of points from an existing point object, based on slope/grade to a given direction and distance. (page 484) |
| CreatePointSlopeGradeElev | Creates a specified number of points from an existing point object, based on slope/grade to a given direction and elevation. (page 485) |
| CreatePoints | Displays the Create Points dialog box. (page 427) |
| CreatePointStationOffset | Creates points on an alignment at a specified station and offset. (page 459) |
| CreatePtAlongLnCurveSpir | Creates points along a line, feature line, lot line, or arc at a given distance from an end point. (page 433) |
| CreatePtAlongPolyContour | Creates points along a polyline or surface contour using elevations from a surface. (page 485) |
| CreatePtIncrementalDist | Interpolates point objects between two points based on an incremental distance from the first point. (page 480) |
| CreatePtIncrmntlElev | Interpolates point objects between two points based on an incremental elevation from the first point. (page 481) |
| CreatePtInterpPerpendic | Creates a point on an object that is either radial or perpendicular to the selected point. (page 452) |
| CreatePtInterpolIntersec | Interpolates point objects between two points, based on an intersection with the selected object. (page 482) |
| CreatePtOnLineCurveSpiral | Creates points at endpoints, PIs, and radius points of objects. (page 434) |

| Command | Description |
|---|---|
| CreatePtPlylineCtrVertMan | Creates points at the vertices of a polyline at a specified elevation. (page 438) |
| CreatePtPlylnCtrVertAuto | Creates points at the vertices of a polyline. (page 439) |
| CreatePtPolyContourVert | Creates points at polyline or surface contour vertices using elevations from a surface. (page 438) |
| CreatePtStationOffsetObj | Creates a point on an object at a specified station and offset. (page 431) |
| EditAllPoints | Modify point properties in the Point Editor. (page 488) |
| EditPointDatum | Changes the elevation of points. (page 492) |
| EditPointNumbers | Changespoint numbers. (page 493) |
| ExportPoints | Exports point data to a text or .mdb file. (page 505) |
| ImportPoints | Imports point data from a text or database file. (page 503) |
| SelectAndEditPointNumbers | Assign a new point number to selected points. (page 493) |
| ShowAllPGChanges | Displays and updates out-of-date point groups. (page 525) |
| ShowDescKeySetsList | Changes the description key set search order. (page 548) |
| ShowGeodeticCalculator | Calculates geodetic information relative to drawing zone and transformation. (page 516) |
| ShowPointFormats | Displays the Point File Format dialog box for creating, copying, or deleting point file formats. (page 2119) |
| ShowPointGroupProperties | Displays the Properties of a selected point group. (page 535) |
| ShowPointGroupsList | Displays all point groups. (page 525) |
| ShowPointSettings | Displays point settings. (page 410) |
| TransferPoints | Transfers points from one file (text or .mdb) to another file (text or .mdb). (page 506) |

# Point Groups

# 16

You can use named collections of points, called point groups, to organize points and to control their appearance in a drawing.

A point group has the following characteristics:

- It has persistent properties you can easily review or change.

- A points list displays the points included in a point group. The point list can be updated automatically. This might be necessary when you change the point group's properties, create new points that match the point group's properties, or erase or modify points that match the point group's properties.

- A point group can be locked to prevent changes within a drawing.

## Understanding Point Groups

Using point groups, you can organize points and control the appearance of points in a drawing.

### Reasons to Use Point Groups

Point groups provide a flexible and convenient way to identify points that share common characteristics or that are used to perform a task, such as creating a surface. You can use point groups to create groupings of points using point number, point name, point elevation, raw (field) or full description, and other characteristics.

Point groups also play a fundamental role in controlling how a point displays in a drawing. If you have a set of points that share common display characteristics, you can use a point group to identify the point style and point label style for all the points in the point group, instead of assigning a point style and a point label style to each individual point. Also, using a point group you can quickly change the style or label style for all the points in a point group at once, instead of changing each point individually.

The point group display order, the point group default styles, and the point group override styles can all affect how a point is drawn. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

### Point Group Properties

A point group is defined by properties which describe the criteria that a point must match to belong to a point group, such as its point number, its name, its raw or full description, or its elevation. Points that match the specified criteria are added to the point group's point list.

You can create a point group either before or after you create the points that belong to it. The point list is maintained dynamically, which means you are notified whenever a change occurs that could affect the point list. For more information, see Out-of-Date Point Groups (page 524).

**NOTE** A point can belong to more than one point group.

## Point Groups Collection (Prospector Tab)

Use the Point Groups collection on the Prospector tab to access point groups in a drawing. As point groups are created, they are listed by their name under the Point Groups collection.

Right-click the Point Groups collection to do the following:

■ Change the point group display order. (page 526)

■ Create a new point group. (page 529)

■ View information about out-of-date point groups before updating them. (page 524)

■ Automatically update all out-of-date point groups.

■ Refresh the display.

Expand the Point Groups collection to display a list of the point groups in the drawing. Right-click a point group to do the following:

■ Edit the point group's properties. (page 535)

■ Edit the points in the point group's point list.

■ Lock or unlock the points in the point group's point list. (page 528)

■ Export the points in the point group. (page 505)

■ Delete the points in the point group.

■ Make a copy of the point group. (page 534)

■ List the changes required to update the point group if it is out-of-date. (page 524)

■ Update the point group if it is out-of-date.

■ Delete the point group. (page 536)

■ Lock or unlock the point group in the drawing. (page 528)

■ Refresh the display.

## The _All Points Point Group

Use the _All Points point group to view a list of all the points in a drawing.

The _All Points point group is created automatically when you create a drawing. It is listed on the Prospector tab under the Point Groups collection, with the other drawing point groups.

When you create a drawing point, it is added to the _All Points point group point list. When you delete a drawing point, it is removed from the _All Points point group point list. A point can belong to other point groups in the drawing, but it is always a member of the _All Points point group.

The _All Points point group serves the following purposes:

■ It provides a default display for points that are created without a point style or point label style and do not belong to any other point group. The appearance of a point in the drawing can be controlled both by the point groups it belongs to and the order in which the point groups are displayed. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

■ It provides a complete list of all the points in a drawing.

Because the _All Points point group point list is automatically managed, you cannot change the point group properties using the Point Groups, Raw Desc Matching, Include, Exclude, and Query Builder tabs in the Point Group Properties dialog box. You can, however, change properties, such as styles and overrides using the Information tab and the Overrides tab.

## Point Group Default Styles

Use the Point Group Properties dialog box to specify a default point style and point label style for a point group.

The default point style and point label style for the point group can be viewed and changed in two locations in the point group properties dialog box: on the Information tab and on the Overrides tab.

If you change the default style on one tab, the other tab is updated to reflect the new style. The check box next to the style name on the Overrides tab indicates when the point group default style is used. If the check box is cleared, then the point group default style is used to display the point only if the point does not have an individual point style assigned. If the check box is selected, the point group default style is used to display each point in the point group, regardless of whether it has an individual point style assigned, according to the point group display order. For more information about point group display order, see Changing the Point Group Display Order (page 526).

Assign individual point styles either during point creation using the options in the Create Points dialog box or using description keys. Use the Point Editor to change individual point styles after a point is created.

For more information, see Controlling the Appearance of Points in a Drawing (page 425).

## Using Point Groups to Override Point Properties

Use the Overrides tab on the Point Group Properties dialog box to specify point property override values for the points that belong to the point group.

When a point belongs to a point group, you can override some of its individual properties using point group *overrides*. When you specify an override value, that value is used to display the point in the drawing and appears in drawing point lists. However, the stored value for the point is not changed.

You can use point groups to override the following point properties:

■ raw description

■ elevation

■ point style

■ point label style

The point group display order can affect the override values displayed in labels for raw description and elevation. It can also affect the point style and point label style that can be used to display a point in a drawing. For more information, see Changing the Point Group Display Order (page 526).

**Raw Description Override**

To specify a raw description override, click ⬚ and enter a value or click ⬚ and select or create an external data reference. For more information, see Using External Data References (page 508). To stop using the override, clear the check box.

**Elevation Override**

To specify an elevation override, click ⬚ and enter a value or click ⬚ and select or create an external data reference. For more information, see Using External Data References (page 508). To stop using the override, clear the check box.

**User-Defined Property Override**

To specify a User-Defined Property override, click ⬚ and cycle through and select ⬚ . Click in the row and select a user-defined property from the list. For more information, see Assigning User-Defined Properties to Point Groups (page 418). To stop using the override, clear the check box.

**Point Style and Point Label Style Overrides**

Use the override point style and point label style to display all the points that belong to the point group. For more information, see Point Group Default Styles (page 523) and Controlling the Appearance of Points in a Drawing (page 425).

# Managing Point Groups

You can manage and update point groups within a drawing.

To manage point groups within a drawing, you can:

- Identify and update out-of-date drawing point groups.

- Lock a point group or the points in a point group's point list.

- Change the point group display order within a drawing.

# Out-of-Date Point Groups

Use the Prospector tab or the Point Groups dialog box to identify out-of-date point groups.

Use the Prospector tree to list the point groups in a drawing. An out-of-date point group has ⬚ next to it in the Prospector tree, in the point group list view, and in the Point Group dialog box. When a point group is out-of-date, the points in the point list no longer match the criteria specified on the tabs in the Point Group Properties dialog box.

**NOTE** Out-of-date icons are displayed only if the Toolspace tree modifier icon display is active. For more information, see Drawing Item Modifier Icons (page 141).

A point group's point list may be out-of-date when one or more of the following occurs:

- You changed a point property (such as raw description or elevation) of a point belonging to the point group, so that the point no longer meets the criteria for being included in the point group.

- You created new points that match the point group's properties.

- You deleted points that belonged to the point group.

■ A point group that is included in the point group (using the Point Groups tab) became out-of-date due to one of the above causes.

For information about updating out-of-date point groups, see Updating All Out-of-Date Point Groups (page 525) and Updating a Single Out-of-Date Point Group (page 526).

## Updating All Out-of-Date Point Groups

Use the Point Groups dialog box to update all out-of-date point groups at once.

A point group is out-of-date when the points in the point list do not match the point group's properties. For more information, see Out-of-Date Point Groups (page 524).

An out-of-date point group has ![icon] next to it in the Prospector tree, the point group list view, or the Point Group dialog box.

**NOTE** Out-of-date icons are displayed only if the Toolspace tree modifier icon display is active. For more information, see Drawing Item Modifier Icons (page 141).

Before you update the point groups, you can use the Point Groups dialog box to view a list of the points that must be added to or removed from the out-of-date point groups to bring them up to date.

**To update all out-of-date point groups**

1 Click COGO Point tab ➤ Modify panel ➤ Edit Point Group List ![icon] .

   The Point Groups dialog box is displayed. Point groups marked with ![icon] are out-of-date.

2 In the Point Groups (page 2118) dialog box, click ![icon] to display information about the out-of-date point groups.

3 In the Point Group Changes (page 2119) dialog box, click ![icon] to update all out-of-date point groups.

4 Click Close in the Point Group Changes dialog box.

5 Click OK in the Point Groups dialog box.

### Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Edit Point Group List ![icon]

Menu

Points ➤ Edit ➤ Point Groups

Toolspace Shortcut Menu

Prospector tab: right-click Point Groups collection ➤ Properties

Command Line

ShowPointGroupsList, ShowAllPGChanges

Dialog Box

  Point Groups (page 2118)

## Updating a Single Out-of-Date Point Group

Use the Prospector tab to update an out-of-date point group.

An out-of-date point group has  ![icon]  next to it on the Prospector tab.

---

**NOTE** Out-of-date icons are displayed only if the Toolspace tree modifier icon display is active. For more information, see Drawing Item Modifier Icons (page 141).

---

A point group is out-of-date when the points in the point list do not match the point group's properties. For more information, see Out-of-Date Point Groups (page 524).

**To update a single out-of-date point group**

1   In Toolspace, on the Prospector tab, right-click a point group that is out-of-date. Click Show Changes.
    The Point Group Changes (page 2119) dialog box displays a list of the changes required to bring the point group up to date.

    ---

    **TIP** If you do not want to review the changes required to bring the point group up to date, click Update on the shortcut menu instead of Show Changes.

    ---

2   Click ![icon] to update the point group.

3   Click OK.

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab: right-click Point Groups item ➤ Show Changes

Dialog Box

  Point Group Changes (page 2119)

## Changing the Point Group Display Order

Use the point group display order to specify how points that belong to more than one point group are displayed in a drawing.
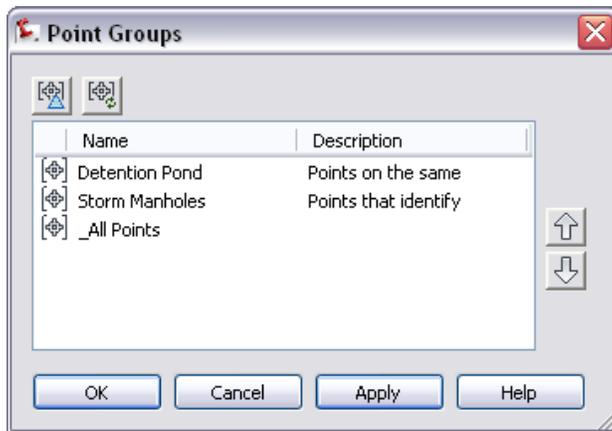
The point group display order for a drawing determines the order in which the point groups are drawn when a drawing is opened or graphics are regenerated. The first (highest) point group in the list is drawn last.

A point is drawn only once each time drawing graphics are regenerated. A point that belongs to more than one point group is drawn by the point group that is highest in the display order, and it is unaffected by the point groups that are lower in the display order. This can determine the point style, point label style, and the layer used to display the point. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

The point group display order is also used to display override values for elevation and description in labels. If a point belongs to more than one point group, the first point group in the display order that draws the point will determine whether override values are used. If the point group has an override for elevation or description, the label is displayed using the override value. If the point group does not have an override, the label is displayed without the override.

**To change the point group display order**

1  In Toolspace, on the Prospector tab, right-click the Point Groups collection. Click Properties.

2  In the Point Groups (page 2118) dialog box, select the point group you want to move in the display order.

3  Click ⬆ or ⬇ .

4  Click OK.

## Quick Reference

Menu

   Points ➤ Edit ➤ Point Groups

Toolspace Shortcut Menu

   Prospector tab: right-click Point Groups collection ➤ Properties

Command Line

   ShowPointGroupsList

Dialog Box

   Point Groups (page 2118)

# Locking and Unlocking Point Groups

Use locking to prevent the properties of a point group from being modified in a drawing.

A locked point group cannot have its properties changed. It cannot be updated (if it is out-of-date), deleted, or overwritten.

A locked point group has  next to it in the Prospector tree and the point group list view. The icon is displayed only if the Toolspace tree drawing state icons are displayed. For more information, see Drawing Item State Icons (page 141).

Locking a point group does not lock the points in the point list. You can lock points individually in the Prospector list view, or you can lock all the points in the point group list at the same time using a single command. For more information, see Locking and Unlocking Point Lists (page 528).

**To lock or unlock a point group**

1   In Toolspace, on the Prospector tab, right-click the point group you want to lock or unlock.

2   Click Lock or Unlock.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Right-click Point Groups item  ➤  Lock or Unlock

# Locking and Unlocking Point Lists

Use point list locking to prevent the points in a point group from being modified in a drawing.

A locked point cannot have its properties changed in the drawing. A locked point has  next to it in point lists such as the Prospector tab Points list view. The icon is displayed only if the Toolspace tree drawing state icons are displayed. For more information, see Drawing Item State Icons (page 141).

Use a single command to lock or unlock all the points in a point group's point list.

**To lock or unlock a point list**

1   In Toolspace, on the Prospector tab, right-click the point group whose point list you want to lock or unlock.

2   Click Lock Points or Unlock Points.

## Quick Reference

Ribbon

Click COGO Point tab ➤ Modify panel ➤ Lock Points 
or
Click COGO Point tab ➤ Modify panel ➤ Unlock Points 

Toolspace Shortcut Menu

Prospector tab: right-click Point Groups item  ➤  Lock Points or Unlock Points

# Specifying Case-Sensitive Description Matching for a Point Group

Specify whether description matching in the Point Group Properties dialog box is case-sensitive.

Case-sensitive matching is not a global setting. You set it separately for each point group.

The Use Case-Sensitive Matching check box on the Query Builder tab of the Point Group Properties dialog box specifies how description matching is performed on the Raw Desc Matching, Include, and Exclude tabs in the Point Group Properties dialog box.

**To specify case-sensitive description matching for a point group**

1 In Toolspace, on Prospector tab, right-click the point group you want to change. Click Properties.

2 In the Point Group Properties dialog box, click the Query Builder (page 2111) tab.

3 To enable or disable case-sensitive matching for raw and full descriptions, select or clear Use Case-Sensitive Matching.

4 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click Point Group item ➤ Properties ➤ Query Builder tab

Command Line

ShowPointGroupProperties

Dialog Box

Query Builder Tab (Point Group Properties) (page 2111)

# Printing Point Groups

Use the Copy To Clipboard command to copy the points in a point group's point list to another application for printing.

To print the points in a point group, click the Prospector tab and then click the point group to display the point list in a list view. Right-click the list view. Click Copy To Clipboard to copy the list view into a file you can print. For more information, see Copying Items from a List View (page 86).

# Creating Point Groups

You create a point group by defining its properties.

Point groups organize and identify points that share common characteristics. Use point groups to assign point styles and point label styles for all the points in a point group instead of assigning styles individually.

Use the tabs on the Point Group Properties dialog box to define the properties of a point group. The tabs you use depend on which method you use to define a point group:

■ The Basic Method. Use this method to easily define the properties for many point groups. Use the Point Groups, Raw Desc Matching, Include, and the Exclude tabs of the Point Group Properties dialog box. Also, use this method when you want to identify points that belong to a point group by selecting points in a drawing.

■ The Query Builder Method. Use this advanced method to create point groups by combining expressions, which allows you to use a single tab, the Query Builder tab, to build a point group based on combinations of point properties or point groups. When you create a point group using the query builder method, the Point Groups, Raw Desc Matching, Include, and Exclude tabs are not available.

---

**NOTE** You should understand how to create a point group using the basic method before using the query builder.

---

After you create a point group, do not try to interchange the two methods to make changes to a point group. You can create a point group using the basic method and then change the point group properties using the Query Builder tab. However, you will lose the changes you made on the Query Builder tab if you then change the point group using the Point Groups tab, the Raw Desc Matching tab, the Include tab, or the Exclude tab.

You can always access the Information tab, the Overrides tab, the Point List tab, and the Summary tab regardless of the method used to define or change the point group's properties.

Because the list of points that belong to a point group (the point list) is updated dynamically, you can create point groups either before or after you create points in the drawing. If you create the points first, and then create the point group, the point list of the newly created point group is immediately updated with the existing points that match the criteria for inclusion in the point group. If you create a point group before you create points that qualify for inclusion in the point group, the point group is marked as out-of-date on the Prospector tab as points are created. You can easily update the point list to include the new points. For more information, see Out-of-Date Point Groups (page 524).

Also, you can create a point group when you import points. In the Import Points dialog box, specify the name of the point group to be created.

## Creating a Point Group Using the Basic Method

Use the basic method to easily define properties for point groups.

Include points in point groups by specifying point descriptions, point numbers, elevations, and point names. You can also include points in a point group by specifying the name of one or more other point groups or by using raw description matching.

**To create a point group using the basic method**

1 Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Point Group [⊕] .

2 In the Point Group Properties dialog box, on the Information (page 2108) tab, enter a name, description, default styles, and a layer for the point group.

3 To include points from other point groups, click the Point Groups (page 2108) tab and select the check box next to any point group you want to include.

4 To include points using raw descriptions, click the Raw Desc Matching (page 2109) tab and select the check box next to the raw descriptions you want to match.

  All points in the drawing are searched, and points with a raw description that matches a raw description specified on the tab are included in the point group.

5 To specify criteria that explicitly includes points in the point group, click the Include (page 2109) tab and define the criteria that a point must match to be included in the point group.

6 To specify criteria that explicitly excludes points from the point group, click the Exclude (page 2110) tab and define the criteria that a point must match to be excluded from the point group.

7 To specify overrides for points in the point group, click the Overrides (page 2113) tab.

8 To view the points in the point group's point list, click the Point List (page 2114) tab.

**9** To view a summary of the point group's properties, click the Summary (page 2114) tab.

**10** Click OK.

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Point Group 

Menu

Points ➤ Create Point Group

Toolspace Shortcut Menu

Prospector tab: Right-click Point Groups collection ➤ New

Command Line

CreatePointGroup

Dialog Box

Point Group Properties (page 2108)

## Creating a Point Group Using the Query Builder

Use the query builder to create point groups by combining expressions.

The Query Builder tab of the Point Group Properties dialog box consolidates many of the options available on the Point Groups, Raw Desc Matching, Include, and Exclude tabs, and allows you to define a complex point group by combining expressions.

For example, to use the basic method to create a point group that includes both the points with a raw description that begins with FLOW and the points in the point group named Flow Line, you would need to use two tabs, the Include tab and the Point Groups tab. You can create the same point group using only the Query Builder tab. The following illustrates the Query Builder tab with a query that specifies that any point that either has a raw description that begins with FLOW or is a member of the Flow Line point group is included in the point group:

| Set Operator | ( | Property | Operator | Value |
|---|---|---|---|---|
| | ( | Raw Description | = | FLOW* |
| OR | | Point Group | = | * |
| | | | | |

The contents of the entire grid on the Query Builder tab is called a *query*. Each row in the query builder is an *expression*. You create a query by combining expressions using the set operators AND, OR, and NOT. For more information about building queries, see Understanding Point Group Queries (page 533).

**NOTE** Since the query builder is an advanced tool for creating point groups, it is recommended that you master point group creation using the basic method before you attempt to use the query builder.

When you first click the Query Builder tab, it displays a starting query that reflects the current contents of the Point Groups, Raw Desc Matching, Include, and Exclude tabs. After you understand how to create a point group using the basic method, you can learn how to build queries by doing the following:

■ Create a point group using the basic method.

■ In the Point Group Properties dialog box, click the Query Builder tab.

■ Examine the starting query, which is created from the information you specified on the Point Groups, Raw Desc Matching, Include, and Exclude tabs.

■ Make changes to the options on one of the above tabs, then return to the Query Builder tab and notice how the changes affected the query.

The Modify Query check box on the Query Builder tab determines which tabs in the Point Group Properties dialog box are active. When this check box is cleared, the Query Builder tab is inactive, and you can change the point group properties using the Point Groups, Raw Desc Matching, Include, and Exclude tabs. When the check box is selected, the Query Builder tab is active, you can change the query, the Point Groups, Raw Desc Matching, Include, and Exclude tabs are inactive.

In general, when you change the properties of a point group, you should set the state of the Modify Query check box once and leave it. Every time you select or clear the Modify Query check box, you can lose information on some tabs in the Point Group Properties dialog box. For example, if you change the query, and then clear the Modify Query check box to activate the other tabs in the dialog box, you lose the changes you made to the query.

**To create a point group using the query builder**

1  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Point Group .

2  In the Point Group Properties dialog box, on the Information (page 2108) tab, enter a name, description, default styles, and a layer for the point group.

3  Click the Query Builder (page 2111) tab.

4  On the Query Builder tab, select Modify Query to activate the query builder.

   A starting query is created based on the contents of the Point Groups, Raw Desc Matching, Include, and Exclude tabs, and those tabs become inactive. If you are creating a new point group, the starting query is empty.

5  Each row in the query builder contains an expression. Do one of the following:

   ■ To create a new expression, activate the first row in the grid by right-clicking it and clicking Insert Row.

   ■ To modify an existing expression in the query or an existing empty row, activate the row containing the expression by clicking in it.

   ■ To delete an expression, click in the row containing the expression and press Delete.

6  Click in the Set Operator column and select a set operator from the list.

7  Click in the "(" column to turn the left parenthesis on or off for the expression.

8  Click in the Property column and select a property.

9  Click in the Operator column and select an operator.

10  Click in the Value column and enter a value that is valid for the property you specified in Step 8.

11  Click in the ")" column to turn the right parenthesis on or off for the expression.

12  Repeat Steps 5 through 11 to build the query.

13  When you finish building the query, click Apply.

14  To specify overrides for points in the point group, click the Overrides (page 2113) tab.

15  To view the points in the point group's point list, click the Point List (page 2114) tab.

16  To view a summary of the point group's properties, click the Summary (page 2114) tab.

**17** Click OK.

### Quick Reference

Ribbon

  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Create Point Group 

Menu

  Points ➤ Create Point Group

Toolspace Shortcut Menu

  Prospector tab: right-click Point Groups collection ➤ New

Command Line

  CreatePointGroup

Dialog Box

  Point Group Properties (page 2108)

## Understanding Point Group Queries

You can use the query builder more effectively if you understand the components of a query.

A query consists of expressions that are combined using set operators.

### Expressions

An expression consists of the following three components:

■ property (for example: Point Number, Point Elevation)

■ comparison operators (for example: =, >, <=)

■ value (for example: 4, 20-87, 101.33)

Use properties, comparison operators, and values to create expressions such as the following:

■ Point Number = 1-100

■ Point Elevation > 1200

■ Point Group = Detention Pond

### Set Operators (AND, OR, and NOT)

Use set operators to combine or negate expressions.

For example, if you combine the first two expressions above with the OR set operator, the resulting query is:

(Point Number = 1-100) OR (Point Elevation > 1200)

The above query adds to the point group any point with either a point number between 1 and 100 or an elevation greater than 1200.

If you combine the first two expressions above with the AND set operator, the resulting query is:

(Point Number = 1-100) AND (Point Elevation > 1200)

The above query adds to the point group any point that has both a point number between 1 and 100 and an elevation greater than 1200.

The NOT operator allows you to exclude points using specified criteria. For example, if you preceded the first expression above with the NOT operator, the resulting query is:

NOT (Point Number = 1-100)

The above query excludes points from the point group if the point number falls between 1 and 100.

### Using Parentheses to Combine Expressions

You can use parentheses to combine expressions. Expressions that fall within parentheses are evaluated before they are combined with other expressions.

The following query uses parentheses to combine expressions:

(Raw Description = MON OR Raw Description = IP) AND (Point Elevation > 200)

Because the previous query is evaluated according to the parentheses, first (Raw Description = MON OR Raw Description = IP) is evaluated and then (Point Elevation > 200) is evaluated, then the two results are evaluated together using the AND set operator. The query adds to the point list only points with a raw description of MON or IP that have elevations greater than 200.

### Precedence of Operators

When parentheses are not used to combine expressions, the query is evaluated using only the following precedence of operators. The following lists the precedence of the operators from highest to lowest:

■ NOT

■ comparison operators (<> <= >= =)

■ AND

■ OR

The following query does not use parentheses to combine expressions:

Raw Description = MON OR Raw Description = IP AND Point Elevation > 200

Because the query is evaluated according to the precedence above, Raw Description = IP AND Elevation > 200 is evaluated first, then the results are evaluated with Raw Description = MON. The query adds to the point list any point that has a raw description of IP and an elevation > 200. It also adds to the point list any point with a raw description of MON, regardless of its elevation.

# Creating a New Point Group From an Existing Point Group

Create a new point group based on an existing point group by creating a copy of the existing point group and then changing the new point group's properties.

Point group names must be unique. When you create a point group by creating a copy, a new point group name is created based on the original point group name.

Change the name of a copied point group on the Information tab of the Point Group Properties dialog box.

### To create a new point group from an existing point group

1  In Toolspace, on the Prospector tab, right-click the point group you want to copy. Click Copy.

2  On the Prospector tab, right-click the new, copied point group. Click Properties.

**3** In the Point Group Properties (page 2108) dialog box, on the Information tab, change the name of the new, copied point group.

**4** Change the point group's properties using either the basic method or the query builder:

■ See Creating a Point Group Using the Basic Method (page 530).

■ See Creating a Point Group Using the Query Builder (page 531).

---

**NOTE** You must use the query builder to change the point group's properties if the Point Groups, Raw Desc Matching, Include, and Exclude tabs are inactive

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### Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click Point Groups item ➤ Copy

# Creating a New Point Group From the Import Survey Data Wizard

A point group is created each time you import a point file using the Import Survey Data wizard.

The point group name defaults to the name of the file you import.

For more information, see Import Survey Data Wizard (page 239).

# Editing Point Groups

You can change the properties of existing point groups or delete a point group.

## Changing the Properties of a Point Group

Use the Point Group Properties dialog box to change the properties of a point group.

The Modify Query check box on the Query Builder tab controls which tabs are active in the Point Group Properties dialog box.

If the Modify Query check box is selected, the Point Groups, Raw Desc Matching, Include, and Exclude tabs are inactive. If the Modify Query check box is cleared, those tabs are active, and you can use them to change the properties of the point group.

**To change the properties of a point group**

**1** In Toolspace, on the Prospector tab, right-click the point group you want to change. Click Properties.

**2** In the Point Group Properties (page 2108) dialog box, use the Information (page 2108) tab to change the name, description, default styles, or layer for the point group.

**3** Change the point group's properties using either the basic method or the query builder:

■ See Creating a Point Group Using the Basic Method (page 530).

■ See Creating a Point Group Using the Query Builder (page 531).

### Quick Reference

## Changing Point Group Properties by Dragging Points into the Point Group

Include drawing points in a drawing point group using the Prospector tree drag-and-drop capability.

The destination point group must be visible in the Prospector tree. The drawing point you want to drag into the point group must be displayed in a list view.

**NOTE** To display all the points in the drawing in a list view, click the _All Points point group.

In the list view, select one or more points. Hold down the left mouse button, and drag the mouse to the

point group name in the Prospector tree. When the cursor display changes to        , release the mouse button
to drop the points into the point group at the tip of the cursor arrow.

The points are added to the point group's point list. The Include tab of the point group's Point Group
Properties dialog box is modified to include the point number of the points.

## Deleting a Point Group

Use the Prospector tree to delete a point group from a drawing.

Deleting a point group does not delete the points contained in the point group's point list. The points remain
in the drawing.

You cannot delete the _All Points point group from a drawing.

**To delete a point group**

1  In Toolspace, on the Prospector tab, right-click the point group you want to delete.

2  Click Delete.

### Quick Reference

# Description Keys

# 17

You can use description keys to automatically control point appearance and some point properties when creating or importing a point into a drawing.

Use description keys to automatically do the following when the drawing point is created:

■ Specify a point style, label style, and a layer for the point.

■ Specify scale and rotation of the point symbol.

■ Translate the raw description for the point into a full description.

## Understanding Description Keys

Before using description keys to create points, you should understand the contents of a description key and how the raw description for a point is used to match a description key.

### Overview

Use description keys to automatically control some drawing point properties, such as the appearance of a point in the drawing, when you create or import points. Before you create drawing points using description keys, create a series of description keys. Then, when you create or import a drawing point, the raw description for the point specifies which description key is used to create the point in the drawing. The properties defined for that description key are applied to the point as it is added to the drawing.

The following are sample description keys, showing code, point style, point label style, format, and layer properties:

| Code | Style | Point Label Style | Format | Layer |
|------|-------|-------------------|--------|-------|
| DRIV | ☑ <default> | ☑ Standard | DRIV $1 | ☑ V-NO |
| FLOV | ☑ <default> | ☑ Standard | $* | ☑ V-NO |
| GRNE | ☐ <default> | ☑ <default> | GROUNC | ☐ |
| LSCP | ☐ <default> | ☑ <default> | LANDSC/ | ☐ |
| MHF' | ☐ <default> | ☑ <default> | PMH | ☐ |

Each description key in a drawing is defined by its properties. Both the code property and the format properties are required in a description key.

Code property. Used during description key matching, for example, if the code matches the raw description for the point, then the properties specified in that description key are applied to the point when it is created.

A description key code can contain characters, called wild cards, that expand the matching capabilities of the description key. For more information, see Description Key Code (page 540).

Format property. Translates the raw description for a point into a full description. The default is $*, which indicates that the full description is the same as the raw description.

Optional description key properties include:

■ point style

■ point label style

■ layer

■ point symbol scaling information

■ point symbol rotation information

### Sample Raw Descriptions

The raw description for a point is used to determine whether a point matches a description key. The following are sample raw descriptions:

MONA U_POLE 1078 TREE OAK 5

The raw description can consist of as many as 10 alphanumeric elements separated by spaces. The leading element in the raw description (MONA, U_POLE, and TREE in the preceding sample) is compared to description key codes during description key matching. The remaining elements, called parameters, can be used to translate the raw description into a more readable full description and to rotate or scale the point symbol in the drawing.

### Description Key Matching During the Creation of Drawing Points

The description key matching process compares the leading element in the raw description of the point you are creating to each of the description key codes in the description keys contained in the drawing. The process ends either when a match is found or when all the description keys in the drawing have been searched.

If the leading element in the raw description for the point matches a description key code, the point style, the point label style, and the layer specified in the description key are used to create the drawing point. The raw description you entered is translated into a full description using the format specified in the description key. If the raw description includes scaling and rotation information, the point symbol is scaled and rotated as specified.

**NOTE** Description key matching is case-sensitive, which means that an upper case letter does not match a lower case occurrence of the same letter. For example, the raw description "TREE" would match the description key code "TREE." It would not match "Tree" or "tree."

### Accessing Description Keys

Description keys are stored in sets within a drawing. All the description key sets available in a drawing are listed in the Settings tree in the Description Key Sets collection.

# Description Key Sets Collection (Settings Tree)

You can use the Description Key Sets collection in the Settings tree to create and manage description key sets.

In the Settings tree, expand the Point collection and then expand the Description Key Sets collection to list the description keys in the drawing.

Right-click the Description Key Sets collection to:

■ [Specify the order in which the description key sets are searched during description key matching](#) (page 548).

■ [Create a new description key set](#) (page 547).

■ Refresh the Settings tree.

Right-click a description key set to:

■ [Display the name and description of the description key set](#) (page 2127).

■ [Edit the description keys in the description key set](#) (page 556).

■ [Copy the description key set](#) (page 548).

■ [Delete the description key set](#) (page 549).

■ Refresh the Settings tree.

# Description Key Settings

Control whether description keys are matched when points are created or imported, and whether the elements in a raw description are interpreted as parameters.

## Activating Description Key Matching

You can control whether description keys are matched when you create or import points.

When you create or import points, clear the Disable Description Keys check box in the Create Points dialog box if you want description key matching to occur.

**NOTE** If the Disable Description Keys setting in the Create Points dialog box is changed, then the Disable Description Keys in the Points Creation settings is changed to the same state if the Save Command Changes To Settings in the Drawing Settings is set to No. For more information about changing the Disable Description Key setting, see [Editing the Default Point Creation Settings](#) (page 415).

If description key matching is active and the raw description for the point matches a description key, the point is created with the point style, point label style, and layer specified in the description key. If a format is specified, the raw description is translated into a full description. If rotation or scaling parameters are specified in the description key, the point symbol is rotated or scaled. For a more information, see [Using Description Keys When Creating Points](#) (page 545) and [Using Description Keys When Importing Points from a File](#) (page 545).

## Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation

You can use the Point Settings dialog box to control whether or not raw descriptions are interpreted as parameters during point creation.

Use the Match On Description Parameters ($1, $2, etc.) setting to specify whether or not parameters are used during description key matching to rotate and scale the point symbol.

■ True. Indicates that parameters are used during description key matching to scale or rotate point symbols. Because spaces are used to delimit description parameters, spaces in description key codes are not recognized. Spaces should not be used when this option is selected.

- False. Indicates that parameters are not used during description key matching to scale and rotate point symbols. Spaces in description key codes are recognized.

**To control whether raw description values are interpreted as parameters during point creation**

1   In Toolspace, on the Settings tab, expand the Commands collection under the Point collection.

2   Right-click CreatePoints. Click Edit Command Settings.

3   In the Edit Point Settings  (page 2103)dialog box, expand the Points Creation collection. For more information, see Editing the Default Point Creation Settings (page 415).

4   For Match On Description Parameters ($1, $2, etc.), select either True or False to specify whether or not raw descriptions contain parameters.

5   Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click command name in object Commands collection  ➤  Edit Command Settings

Dialog Box

Edit Command Settings (page 1809)

# Description Key Properties

Each description key property performs a distinct function during description key matching and point creation.

Description key properties specify how a description key is matched and how the raw description for a point is translated to a full description. They can also be used to specify a point style, a point label style, and a layer for a point that matches the description key, and scaling and rotation values for the point symbol.

For a definition of all description key properties and instructions for editing them using the DescKey Editor, see Description Key Editor (page 2128).

# Description Key Code

Use the code property of a description key to specify which raw descriptions match the description key.

A description key consists of a code and a set of additional properties. When the code is matched, the other properties specified in the description key are applied to the point when it is created.

The following are sample description key codes:

DRIVS*
FLOW*
GRND*
LSCP*
MHE*
MHST*

You can either specify a simple string for a description key code or use wild card characters, such as an asterisk (**), to expand the matching capabilities of description keys.

For example, all your benchmark points might have the BM prefix in their raw descriptions, such as BM-1 and BM-2. When you add benchmark points to a drawing in AutoCAD Civil, you might want them all to

be created with the same point style, point label style, and description. To do this, you can create a description key with a code of BM*. (The asterisk means that the code will match all raw descriptions that begin with BM.) Then, all points with a raw description that begins with BM are created using the properties specified in that description key.

The following table shows wild card characters you can use in a description key code:

| Character | Function in Description Key Code |
| --- | --- |
| # (pound) | Matches any single numeric digit. |
| @ (at) | Matches any alphabetic character. |
| . (period) | Matches any non-alphanumeric character. |
| * (asterisk) | Matches any string and can be used anywhere in the search string. |
| ? (question mark) | Matches any single character, for example, ?BC matches ABC and 3BC. |
| ~ (tilde) | Matches anything but the pattern, for example, ~*AB* matches all strings that don't contain AB. |
| [...] (brackets) | Matches any one of the characters enclosed, for example, [AB]C matches AC and BC. |
| [~...] | Matches any character not enclosed, for example, [~AB]C matches XC but not AC. |
| - (hyphen) | Inside brackets, specifies a range for a single character, for example [A-G]C matches AC, BC, through GC, but not HC. |
| ' (reverse quote) | Reads the next characters exactly, for example, '*AB matches *AB. |

The following table shows examples of description key codes containing commonly used wild card characters:

| Description Key Code | Function |
| --- | --- |
| T# | Matches descriptions that consist of T followed by one digit, such as T1, T2, through T9. |
| STA# | Matches descriptions that consist of STA followed by one digit, such as STA1, STA2, through STA9. It does not match the description STA, since STA is not followed by a digit. |
| T## | Matches descriptions that consist of T followed by 2 digits, such as T01 through T99. |
| 1@ | Matches descriptions that consist of 1 followed by an alphabetic character, such as 1A, 1B, and 1C. It does not match the description 1, since 1 is not followed by an alphabetic character. |
| T. | Matches descriptions that consist of T followed by a single non-alphanumeric character, such as T- and T+. |

| Description Key Code | Function |
|---|---|
| T* | Matches descriptions that consist of T followed by any number of characters, such as T1, TOPO, T-2, and TREE. |

**NOTE** Description key matching is case-sensitive, which means that an upper case letter does not match a lower case occurrence of the same letter. For example, the raw description "TREE" would match the description key code "TREE." It would not match "Tree" or "tree."

# Description Key Format

Use the format property of a description key to create a full description for a point that matches a description key.

The following are sample description key formats:

```
DRIV $1
$*
GROUND
LANDSCAPE
PMH
DMH
```

The raw description for a point is always stored with the point and often corresponds to the point description entered by a surveyor in the field. Sometimes, for example, when you are labeling a point in a drawing, the raw description for the point does not contain information in the format you want. You can use the format property to create a full description that contains a more readable description of the point.

### Simple Methods of Specifying Full Descriptions

Define a standard full description for points that match a description key by specifying the desired description in the format property. For example, if you want all points that match a specific description key to have a full description of MONUMENT, enter MONUMENT for the format. Then, points that match the description key will be created with the full description MONUMENT.

If you want the full description to be the same as the raw description for all points that match the description key, enter $* for the format.

### Using Parameters to Translate a Raw Description into a Full Description

Create a readable full description from a raw description by either changing the order of the elements in the raw description or adding descriptive text. To do this, you must define a format that translates the raw description into a full description.

The raw description for a point consists of a list of elements separated by spaces, for example, TREE OAK 7. To re-order the elements in a raw description, you must reference the individual elements. When creating a format, you reference the leading element in a raw description using the symbol $0. You reference the remaining elements, which are referred to as parameters, using a $ followed by a number (1-9) that specifies the position for the parameter in the raw description.

For example, in the raw description TREE OAK 7, the leading element in the raw description, TREE, is referenced using the symbol $0. The first parameter, OAK, is referenced using the symbol $1. The second parameter, 7, is referenced using the symbol $2.

When a description key match occurs, the raw description is translated into a full description using the format, which references the elements in the raw description.

**NOTE** The leading element in a raw description is not referred to as a parameter. "Parameter" refers to the individual elements that follow the leading element in a raw description.

**Example**

You are creating a point, and for the raw description, you specify TREE OAK 7. One of your description key sets contains a description key with the code TREE and the format "$2 inch $1 tree."

The leading element in the raw description, TREE, matches the code in the description key, which is also TREE. The format specified in the description key, "$2 inch $1 tree," is used to translate the raw description into a full description for the point.

The $2 in the format refers to the second parameter in the raw description, which is 7. The $1 in the format refers to the first parameter in the raw description, which is OAK. The full description is created by substituting 7 for $2 and OAK for $1 in the format "$2 inch $1 tree." The text in the format ("inch" and "tree") is left unchanged. After the substitutions are made, the translated full description is "7 inch OAK tree."

The following is a chart of description key parameter substitution codes, including the code to use if you want the raw description to be used for the full description:

| Use this Code... | To do this in a format... | Example |
|---|---|---|
| $0 | Reference the leading element in the raw description. | **TREE** Oak 7 |
| $1 | Reference the first parameter in the raw description. | TREE **Oak** 7 |
| $2 | Reference the second parameter in the raw description. | TREE Oak **7** |
| $3 | Reference the third parameter in the raw description. | TREE Oak 7 **24** |
| $4, $5, through $9 | Reference the fourth parameter, the fifth parameter, through the ninth parameter. | |
| $+ | Reference all the parameters in the raw description after the leading element. | TREE **Oak 7** |
| $* | Reference all the parameters in the raw description and also the leading element. | **TREE Oak 7** |
| $$ | Insert a single $ into the description. | If you used $$200.00 as part of the format, then that part of the translated full description would be $200.00. |
| $* | Use the point's raw description for the full description. | If you used the raw descriptions UP-1, UP-2, and UP-3, and you want to use these descriptions for the full description, then use the $* as the format. |

# Specifying Point Symbol Scaling and Rotation Using Description Key Parameters

Use parameters to specify the point symbol scale or rotation value using the raw description for the point when a point matches a description key.

Parameters are elements that are included in the raw description for a point. For example, the raw description TREE OAK 7 consists of a leading element, TREE, followed by two parameters, OAK and 7, separated by spaces.

### Specifying Scale Using a Parameter

If you want the point symbol scale to be read from the raw description for the point when a description key match is found, use the scale parameter property. In the previous example, OAK is parameter 1, and 7 is parameter 2. If you wanted the description key to take the symbol scale value from the second parameter (7) you would specify Parameter 2 as the scale parameter.

For a complete description of all the options available for scaling a point symbol using a description key, see the Description Key Editor (page 2128).

### Specifying Rotation Using a Parameter

If you want the point symbol rotation to be read from the raw description for the point when a description key match is found, use the rotate parameter property. For example, a raw description Hyd 45 A consists of the leading element Hyd followed by two parameters: 45 is parameter 1, and A is parameter 2. If you wanted the description key to use 45 to rotate the point symbol, you would specify parameter 1 as the rotate parameter.

For a complete description of all the options available for rotating a point symbol using a description key, see the Description Key Editor (page 2128).

**NOTE** You can also use parameters in the raw description to create a full description by using a description key format. For more information, see Description Key Format (page 542).

# Description Key Properties That Are Stored with a Point

Some description key properties are stored with the point in the drawing.

When you create a point and description key matching is active, and a match is found, the point is created using the properties specified in the description key. The following description key properties are stored with the point, and can be changed for an individual point to alter its appearance in the drawing:

- Format
- X-Y Scale
- Z Scale
- Rotation

Use the Point Editor to change these properties.

# Managing Description Keys

You can use the Settings tree to manage description keys.

Description keys are organized into sets. The description key sets for a drawing are listed in the Settings tree under the Description Key Sets collection, which is found under the Point collection.

Each description key set contains one or more description keys. To view or edit the description keys contained within a set, use the DescKey Editor. For more information, see Editing Description Keys (page 556).

When description key matching is active and a drawing point is created, each description key code in every description key set in the drawing is searched for a match. You can control the order in which the description key sets are searched.

Include description key sets in a drawing template so they can be used in multiple drawings and shared with other people. For more information, see Drawing Templates (page 109).

## Using Description Keys When Creating Points

Use description keys when you create drawing points using the Create Points command.

When you create a drawing point and description key matching is active, the raw description you enter at the prompt initiates a search. The leading element in the raw description is compared to each description key code in every description key set in the drawing until either a match is found or all description key sets have been searched.

If a match is found, the point style, the point label style, and the layer specified in the description key are used to create the point. The raw description you entered is translated into a full description using the format specified in the description key. If the raw description includes scaling and rotation information, the point symbol is scaled and rotated as specified.

If a description key match is not found, the point is created without a point style or point label style. The point is displayed using point group styles. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

For more information about working with description keys, see Understanding Description Keys (page 537).

**To use description keys when creating points**

1  Create one or more description key sets containing the description keys you want to use to create the points. For more information, see Creating Description Keys (page 550).

2  Specify the description key sets search order. For more information, see Changing the Description Key Sets Search Order (page 548).

3  For Prompt For Descriptions Points Creation select Manual. For more information, see Editing the Default Point Creation Settings (page 415).

4  Specify the setting that indicates whether your description keys use parameters. See Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation (page 539).

5  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools ⬚ .

6  Expand the Create Points (page 2117) dialog box. Expand Points Creation and set the Disable Description Keysvalue to True.

7  Click a command on the toolbar. Follow the prompts to create a point.

8  When prompted for a description, enter the leading element of a raw description that matches the code specified in one of your description keys. If your description key requires parameters to complete the full description or to rotate or translate the symbol, enter the parameters, separated by spaces.

   The point is drawn using the properties specified in the description key, such as point style, point label style, and layer.

## Using Description Keys When Importing Points from a File

You can use description keys when you import points into a drawing.

Create drawing points by importing point data from a point data file, which can be either an ASCII text file or a Microsoft® Access database (*.mdb*) file. For more information about creating points by importing, see Importing Point Data (page 503).

When you import points from a point data file and description key matching is active, the raw descriptions in the point data file are used for matching. For each point that is imported, the leading element in its raw description is compared to each description key code in every description key set in the drawing until either a match is found or all description key sets have been searched.

If a match is found, the point style, the point label style, and the layer specified in the description key are used to create the point in the drawing. The raw description for the point is translated into a full description using the format specified in the description key. If the raw description includes scaling and rotation information, the point symbol is scaled and rotated as specified.

If a description key match is not found, the point is created without a point style or point label style, and the point displays using point group styles. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

For more information about working with description keys, see Understanding Description Keys (page 537).

**To use description keys when importing points from a file**

1   Create one or more description key sets containing the description keys you want to use when importing the points. For more information, see Creating Description Keys (page 550).

2   Specify the description key sets search order. For more information, see Changing the Description Key Sets Search Order (page 548).

3   Specify the setting that indicates whether or not your description keys use parameters. For more information, see Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation (page 539).

4   Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 .

5   Expand the Create Points (page 2117) dialog box.

6   Expand Points Creation and set the Disable Description Keysvalue to True. On the Points Creation toolbar, clickImport Points ⬆ .

7   In the Format list, specify a format name.

8   Import the points. For more information, see Importing Point Data (page 503).

    Any point with a raw description that matches a description key is created in the drawing using the properties specified in the description key, such as point style, point label style, and layer.

## Using Description Keys to Include Points in a Point Group

Use description key codes to include points in a point group.

You can include points in a point group based on description key matching. Use the Raw Desc Matching tab in the Point Group Properties dialog box to specify the description keys a point can match to be included in the point group. For more information, see Raw Desc Matching Tab (Point Group Properties Dialog Box) (page 2109).

You can also use either the raw description or the full description for a point to either include or exclude points from a point group. For more information, see Include Tab (Point Group Properties Dialog Box) (page 2109) and Exclude Tab (Point Group Properties Dialog Box) (page 2110).

# Creating a New Description Key Set

Use the Settings tree to create a description key set.

A description key set must have at least one description key in it. When you open a newly created description key set with the DescKey Editor, a starting description key is automatically added to the description key set.

**To create a new description key set**

1  In Toolspace, on the Settings tab, expand the Point collection.

2  Right-click the Description Key Sets collection. Click New.

3  In the Description Key Set (page 2127) dialog box, enter a name and description for the description key set.

4  Click OK.

   The description key set is created.

5  Right-click the new description key set. Click Edit Keys.

6  In the Description Key Editor (page 2128), edit the properties of the starting description key.

7  Add additional description keys to the description key set. For more information, see Creating a New Description Key (page 550).

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click Description Key Sets collection ➤ New

Command Line

  CreateDescKeySet

Dialog Box

  Description Key Set (page 2127)

# Importing a Description Key Set from AutoCAD Land Desktop

Import a description key set from AutoCAD Land Desktop.

You can import description key sets that you created in AutoCAD Land Desktop to use with COGO points.

**To import a description key**

1  Click Insert tab ➤ Import panel ➤ Land Desktop  .

2  In the Import Data From Autodesk Land Desktop Project dialog box, enter the Land Desktop Project Path and the Project name.

3  Select the Description Keys check box and click OK.

### Quick Reference

Click Insert tab ➤ Import panel ➤ Land Desktop

Menu

File menu Import ➤ Import Data From Land Desktop

# Creating a Description Key Set Based on an Existing Description Key Set

Use the Settings tree to copy a description key set.

When you create a copy of a description key set, the new description key set contains a copy of all the description keys in the original set.

**To create a description key set based on an existing description key set**

1  In Toolspace, on the Settings tab, expand the Point collection.

2  Expand the Description Key Sets collection.

3  Right-click the description key set you want to copy. Click Copy.

   A copy of the description key set is created. Its name is based on the name of the existing description key set.

4  Right-click the new description key set. Click Properties.

5  In the Description Key Set (page 2127) dialog box, enter a name and description for the new description key set.

6  Click OK.

7  Right-click the new description key set. Click Edit Keys.

8  In the Description Key Editor (page 2128), edit the description keys in the new description key set.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Description Key Sets item ➤ Copy

# Changing the Description Key Sets Search Order

Specify the order in which description key sets are searched for matching description keys.

The Description Key Sets Search Order dialog box contains a list of description keys. The first description key set on the list is searched first, and the rest of the description key sets are searched sequentially in order, ending with the last description key set on the list.

**To change the search order of description key sets**

1  In Toolspace, on the Settings tab, expand the Point collection.

2  Right-click the Description Key Sets collection. Click Properties.

**3** In the Description Key Sets Search Order (page 2128) dialog box, click the name of a description key set to select it.

**4** Click ⬆ or ⬇ to move the point group up or down in the search order.

**5** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Description Key Sets collection ➤ Properties

Command Line

ShowDescKeySetsList

Dialog Box

Description Key Sets Search Order (page 2128)

## Renaming a Description Key Set

Use the Settings tree to rename a description key set.

The new name does not affect the description keys contained within the set or the position of the description key set within the search order.

**To rename a description key set**

**1** In Toolspace, on the Settings tab, expand the Point collection.

**2** Expand the Description Key Sets collection.

**3** Right-click the description key set you want to rename. Click Properties.

**4** In the Description Key Set (page 2127) dialog box, enter a new name for the description key set.

**5** Optionally, enter a description.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Description Key Sets item ➤ Properties

## Deleting a Description Key Set

Use the Settings tree to delete a description key set.

When you delete a description key set, all the description keys in the description key set are deleted.

For information about deleting an individual description key, see Deleting a Description Key (page 557).

**To delete a description key set**

**1** In Toolspace, on the Settings tab, expand the Point collection.

**2** Expand the Description Key Sets collection.

**3** Right-click the description key set you want to delete. Click Delete.

**4** Click Yes.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Description Key Sets item ➤ Delete

## Printing Description Keys

Use the Copy to Clipboard command to either print a list of the description key sets in a drawing or to print a list of the description keys contained within an individual description key set.

To print a list of the description key sets in a drawing, click the Description Key Sets collection in the Settings tree to display a list view that contains the description key sets in the drawing. Right-click in the list view to display a shortcut menu. Use the Copy to Clipboard command to copy the contents of the list view into a file you can print.

To print a list of the description keys in a description key set, click the description key set in the Settings tree to display a list view that contains the description keys in the description key set. Right-click in the list view to display a shortcut menu. Use the Copy to Clipboard command to copy the contents of the list view into a file you can print.

For more information about the Copy to Clipboard command, see Copying Items from a List View (page 86).

## Creating Description Keys

You can use the DescKey Editor to create new description keys and to copy description keys.

You can also perform the these operations in the Prospector item view. However, the DescKey Editor, because it is a separate window, provides more viewing area and greater flexibility in your use of screen space.

## Creating a New Description Key

Use the DescKey Editor to create a new description key.

A new description key is always created with default properties. After you create the description key, you can change the properties.

**To create a new description key**

**1** In Toolspace, on the Settings tab, expand the Point collection.

**2** Expand the Description Key Sets collection.

**3** Right-click the description key set to which you want to add the description key. Click Edit Keys.

> **NOTE** You can create a new description key set to which you can add the description key. For more information, see Creating a New Description Key Set (page 547).

**4** In the Description Key Editor (page 2128), right-click in the section that contains description keys. Click New.

A new description key is created with default properties.

5   If necessary, display the columns that contain the properties you want to edit. Display and hide the columns in the DescKey Editor as you would in a list view. For more information, see Customizing a List View (page 84).

6   Edit the description key properties as needed. For more information, see Description Key Editor (page 2128).

7   Close the DescKey Editor.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Description Key Sets ➤ <description key set> ➤ right-click in List View ➤ New

# Creating a Description Key Based on an Existing Description Key

Use the DescKey Editor to copy a description key.

Create a copy of a description key within the same description key set. You cannot copy a description key from one description key set to another.

**To create a description key based on an existing description key**

1   In Toolspace, on the Settings tab, expand the Point collection.

2   Expand the Description Key Sets collection.

3   Right-click the description key set to which you want to add the description key. Click Edit Keys.

4   In the Description Key Editor (page 2128), right-click the description key you want to copy. Click Copy.

    A copy of the description key is created.

5   If necessary, display the columns that contain the properties you want to edit in the new description key. Display and hide the columns in the DescKey Editor as you would in a list view. For more information, see Customizing a List View (page 84).

6   Edit the properties of the new description key as needed. For more information, see Description Key Editor (page 2128).

7   Close the DescKey Editor.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Description Key Sets ➤ <description key set> ➤ right-click in List View ➤ Copy

# Example: Creating a Basic Description Key

This example shows how to create and use a basic description key.

In this example, you create a description key that is used to create utility pole points that have raw descriptions with a prefix of UP, for example, UP5A and UP4B. When these points are created in the drawing, they are displayed with the same symbol and layer. Each point maintains its individual description.

This example shows how to use a wild card character in the description key code so that a single description key matches any point whose raw description begins with the prefix UP.

As you work through this example, you might want more information about the contents of the columns in the DescKey Editor. For more information, see Description Key Editor (page 2128).

**Example: Creating a basic description key**

1   Before you begin, create a point style named UP_Symbol to be used to display the utility pole points created in this example. For more information, see Point Styles (page 419).

2   In Toolspace, on the Settings tab, expand the Point collection.

3   Right-click the Description Key Sets collection. Click New.

4   For Name, enter UP_Example. Click OK.

    A description key set named UP_Example is created and is listed in the Settings tree under the Description Key Sets collection.

5   Right-click UP_Example. Click Edit Keys.

    The DescKey Editor is opened. If you cannot see the DescKey Editor, click ⊞ at the top of the Settings tab to display the Panorama window.

    If you cannot see the column headings mentioned in the following steps, use the column heading shortcut menu to display or expand the column headings. You display and hide the columns in the DescKey Editor as you would in a list view. For more information, see Customizing a List View (page 84).

6   In the DescKey Editor, click in the first cell under the Code column heading. Enter UP* for the code.

    The asterisk (*) indicates that the description key will match any raw description that starts with UP, such as UP5A and UP5B.

7   Under the Point Style column heading, select the check box and click in the cell. In the Point Style dialog box, select UP_Symbol. Click OK.

    The UP_Symbol style will be used to display the symbol for points that match the description key you are creating.

8   Under the Point Label Style column heading, make sure that the check box is selected and that the Standard style is specified. The Standard style labels the point with point number, point elevation, and full description.

9   Click in the cell under the Format column heading, and enter $*.

    The dollar sign asterisk ($*) format means that the full description and raw description for the point will be the same.

    If you wanted, you could enter text, such as UTILITY POLE, which would be used as the full description for all points with a raw description beginning with UP, or you could assign a format that creates a full description from the raw description. For more information, see Description Key Format (page 542).

10  Under the Layer column heading, select the check box. Click in the cell and create a layer named PTS_UP.

11  Clear the check boxes under all other column headings.

    A point created using this description key will be placed on the PTS_UP layer.

12  Close the DescKey Editor.

13  If you have multiple description key sets defined, move the UP_Example set to the top of the search order. For more information, see Changing the Description Key Sets Search Order (page 548).

**14** Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools . In the Create Points dialog box, click  to expand the dialog box.

**15** Expand the Points Creation collection and specify the following settings:

■ For the Prompt For Descriptions, select Manual. For more information, see Editing the Default Point Creation Settings (page 415)

■ For the Disable Description Keys, set the value to False.

**16** From the Miscellaneous command list, select Manual.

**17** Pick the location for a new point.

**18** Follow the prompts. When you are prompted for the description, enter UP1A.

> **NOTE** Description key matching is case-sensitive, which means that an upper case letter does not match a lower case occurrence of the same letter. For example, the raw description "UP1A" matches the description key code "UP*", but it would does not match "up*" or "Up*".

**19** The point is drawn using the UP_Symbol point style and the Standard point label style. The point is created on the PTS_UP layer.

## Example: Using Description Key Parameters to Create a Full Description

This example shows how to use parameters in a description key format to translate a raw description into a full description.

> **NOTE** The following paragraphs are intended to help you understand this example. They do not provide a complete overview of description keys. For general information about description keys, see Understanding Description Keys (page 537). Before beginning this example, you should complete the example that shows you how to create a basic description key. For more information, see Example: Creating a Basic Description Key (page 551).

When you create a point, the description you enter at the command line is the raw description for the point. The raw description can consist of one element, such as UP5A, or of more than one element, such as UPNO 104. The leading element in a raw description is compared against description key codes during description key matching. The remaining elements in the raw description are called parameters. You can use parameters to create a full description for all points that match the description key.

In this example, you will create a description key that translate a raw description in the form UPNO <pole number> to a full description in the form U_POLE <pole number>. For example, the field (raw) description UPNO 104 will be translated to the full (drawing) description U_POLE 104.

In the raw description UPNO 104, UPNO is used to match the description key code and 104 is the first parameter. For more information, see Description Key Format (page 542).

**Example: Using description key parameters to create a full description**

**1** In Toolspace, on the Settings tab, right-click the Description Key Sets collection. Click New.

**2** For Name, enter Format_Example. Click OK.

A description key set named Format_Example is created.

**3** Right-click Format_Example. Click Edit Keys.

The DescKey Editor is opened. If you cannot see the DescKey Editor, click  at the top of the Settings tab to display the Panorama window.

If you cannot see the column headings mentioned in the following steps, use the column headings shortcut menu to display or expand the column headings. You display and hide the columns in the DescKey Editor the same way you display and hide the columns in a list view. For more information, see Customizing a List View (page 84).

4   In the DescKey Editor, click in the first cell under the Code column heading, and enter UPNO for the code.

The code will match any raw description whose leading element is UPNO.

5   Under both Point Style and Point Label Style heading columns, select the check boxes. Verify that the Standard style is specified.

**NOTE** The Standard point label style displays a full description for the point.

6   Click in the cell under the Format column heading, and enter U_POLE $1.

$1 refers to the first parameter in the raw description. This format will translate the raw description "UPNO 104" to the full description "U_POLE 104." For more information, see Description Key Format (page 542).

7   Under the Layer column heading, select the check box, click in the cell and create a layer named UP_LAYER. A point created using this description key will be placed on the UP_LAYER layer.

8   Clear the check boxes under all other column headings.

9   Close the DescKey Editor.

10  If you have multiple description key sets defined, move the Format_Example description key set to the top of the search order. For more information, see Changing the Description Key Sets Search Order (page 548).

11  Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 🔧 .

12  In the Create Points dialog box, click ⮟ to expand the dialog box.

13  Expand the Points Creation collection and specify the following settings:
■   For the Prompt For Descriptions, select Manual. For more information, see Editing the Default Point Creation Settings (page 415)

■   For the Match on Description Parameters ($1, $2, etc.)  select True. For more information, see Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation (page 539).

■   For the Disable Description Keys, set the value to False.

14  From the Miscellaneous command list, select Manual.

15  Pick the location for a new point.

16  Follow the prompts. When you are prompted for the description, enter UPNO 104.

The point is drawn using the full description derived from the description key, U_POLE 104.

## Example: Using Description Key Parameters to Format a Full Description and Scale a Point Symbol

This example shows how to use description key parameters to both expand the raw description for a point and to scale a point symbol.

**NOTE** The following paragraphs are intended to help you understand this example. They do not provide a complete overview of description keys. For an overview of description keys, see Understanding Description Keys (page 537). Before beginning this example, you should complete the example that shows you how to create a basic description key. For more information, see Example: Creating a Basic Description Key (page 551).

When you create a point, the description you enter at the command line is the raw description for the point. The raw description can consist of one element, such as TREE, or it can consist of more than one element, such as TREE OAK 7. The leading element in a raw description is compared against description key codes during description key matching. The remaining elements in the raw description are called parameters. You can use parameters to create a full description for a point or to rotate or scale a point.

In this example, you create a description key that translates a raw description with the format TREE <tree type> <tree size> into a full description that reads: <tree size> inch <tree type> tree, and you will scale the point symbol using the <tree size> value. For example, the raw description TREE OAK 7 will be translated to a full description 7 inch OAK tree. The point symbol will be scaled using a value of 7.

In the raw description TREE OAK 7, OAK is the first parameter and 7 is the second parameter. For more information, see Description Key Format (page 542).

**Example: Using parameters to format a full description and scale a point symbol**

1. Before you begin, create a point style named Tree_Symbol which will display the tree points created in this example. For more information, see Creating a Point Style (page 420).

2. In Toolspace, on the Settings tab, right-click the Description Key Sets collection. Click New.

3. For Name, enter Param_Example. Click OK.

   A description key set named Param_Example is created.

4. Right-click Param_Example. Click Edit Keys.

   The DescKey Editor is displayed. If you cannot see the DescKey Editor, click ⊞ at the top of the Settings tab to display the Panorama window.

   If you cannot see the column headings mentioned in the following steps, you may need to use the column headings shortcut menu to display them, or you may need to expand the column headings. You display and hide the columns in the DescKey Editor as you would in a list view. For more information, see Customizing a List View (page 84).

5. In the DescKey Editor, click in the first cell under the Code column heading, and enter TREE for the code.

   The code will match any point raw description whose leading element is TREE.

6. Under the Point Style column heading, select the check box and click in the cell. In the Point Style dialog box, select Tree_Symbol. Click OK.

7. Under the Point Label Style column heading, select the check box. Verify that the Standard point label style is specified.

8. Click in the cell under the Format column heading, and enter $2 inch $1 Tree.

   $2 refers to the second parameter in the raw description. $1 refers to the first parameter in the raw description. This format will translate the raw description "TREE OAK 7" into the full description "7 inch OAK tree."

   For more information, see Description Key Format (page 542).

9. Under the Layer column heading, select the check box. Click in the cell and create a layer named PTS_TREE. A point created using this description key will be placed on the PTS_TREE layer.

10. Under Apply To X-Y, select the check box.

11. Under Apply To Z, select the check box.

**12** Under Scale Parameter, select the check box.

**13** Under Scale Parameter, select the check box. Select Parameter 2.

Parameter 2 refers to the second parameter in the raw description, which is 7 in this example. This specifies that the second parameter in a description contains the value by which the point symbol is scaled.

**14** Clear the check boxes under all other column headings.

**15** Close the DescKey Editor.

**16** If you have multiple description key sets defined, move the Param_Example description key set to the top of the search order. For more information, see Changing the Description Key Sets Search Order (page 548).

**17** Click Home tab ➤ Create Ground Data panel ➤ Points menu ➤ Point Creation Tools 📍 .

**18** In the Create Points dialog box, click ⌄ to expand the dialog box.

**19** Expand the Points Creation collection and specify the following settings:

■ For the Prompt For Descriptions, select Manual. For more information, see Editing the Default Point Creation Settings (page 415)

■ For the Match on Description Parameters ($1, $2, etc.), select True. For more information, see Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation (page 539).

■ For the Disable Description Keys, set the value to False.

**20** From the Miscellaneous command list, select Manual.

**21** Pick the location for a new point.

**22** Follow the prompts. When you are prompted for the description, enter TREE OAK 7.

A point is drawn and labeled with the full description 7 inch OAK tree, and the tree symbol is scaled.

**23** Create another point, and specify TREE OAK 5 for the raw description to see the impact on the full description and the scaled symbol.

# Editing Description Keys

You can use the DescKey Editor to edit and delete description keys.

To edit a description key, you must open the description key set that contains the description key.

The DescKey Editor is displayed as a vista (a tab) in the Panorama window. For more information about working with vistas, see The Panorama Window (page 102).

The DescKey Editor displays the description keys in a grid. Each row in the grid contains a description key. The columns contain the description key properties. You display and hide the columns in the DescKey Editor as you would in a list view. For more information, see Customizing a List View (page 84).

For a complete description of each column in the DescKey Editor, see Description Key Editor (page 2128).

**NOTE** You can edit description keys in the Prospector item view. However, the DescKey Editor, because it is a separate window, provides you with more viewing area and greater flexibility in your use of screen space.

# Opening the Description Key Editor

Use the Settings tree to open the DescKey Editor.

Before you can open the DescKey Editor, you must select a description key set to edit. The DescKey Editor displays all the description keys in the selected description key set.

**NOTE** You can also edit description keys in the Prospector item view. However, the DescKey Editor, because it is a separate window, provides you more viewing area and greater flexibility in your use of screen space.

**To open the Description Key Editor**

1 In Toolspace, on the Settings tab, expand the Description Key Sets collection.

2 Right-click the description key set you want to edit. Click Edit Keys.

3 Use the Description Key Editor (page 2128) to edit the individual description keys.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Description Key Sets item ➤ Edit Keys

Dialog Box

Description Key Editor (page 2128)

# Deleting a Description Key

Use the DescKey Editor to delete a description key from a description key set.

For information about deleting an entire description key set, see Deleting a Description Key Set (page 549).

**To delete a description key**

1 In Toolspace, on the Settings tab, right-click the description key set that contains the description key you want to delete. Click Edit Keys.

2 In the Description Key Editor (page 2128), right-click the description key you want to delete. Click Delete.

3 Close the DescKey Editor.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Description Key Sets ➤ <description key set> ➤ right-click in List View ➤ Delete

# Lines and Curves

# 18

Many commands for creating lines and curves are included in AutoCAD Civil.

## Creating Lines

You can use the AutoCAD Civil line creation commands to draft line geometry prior to defining it as alignments or parcels, for example.

Several of the AutoCAD Civil Line commands are simply the AutoCAD Line combined with an AutoCAD Civil transparent command. For example, when you run the Line By Bearing command, the AutoCAD Line command starts, but the 'BD transparent command also launches so you can immediately enter a bearing value. For more information on transparent commands, see Transparent Commands (page 1581).

## Creating Lines By Selecting a Start and End Point

The Line command on the Home tab ➤ Draw panel launches the AutoCAD LINE command. Use this command to draw a line by selecting start and end points.

**To create a line by selecting a start and end point**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line .

2 Specify the start point.

You can use the pointing device or enter coordinate values on the command line.

3 Complete the first line segment by specifying the endpoint.

To undo the previous line segment during the Line command, enter **U** or click Undo on the toolbar.

4 Specify the endpoints of any additional line segments.

5 Press Enter to end or **C** to close a series of line segments.

6 To start a new line at the endpoint of the last line drawn, start the Line command again and press Enter at the Specify Start Point prompt.

**Quick Reference**

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line ✎

Menu

Lines/Curves menu ➤ Create Lines ➤ Line

Command Line

Line

# Creating Lines by Specifying a Range of Point Numbers

The Line By Point # Range command runs the AutoCAD Line command combined with the 'PN transparent command (page 1590) so you can specify a range of point numbers through which to draw the line.

You can specify a sequence of point numbers by using a comma. For example, the sequence 1-3,7 draws a line between points 1, 2, 3, and 7.
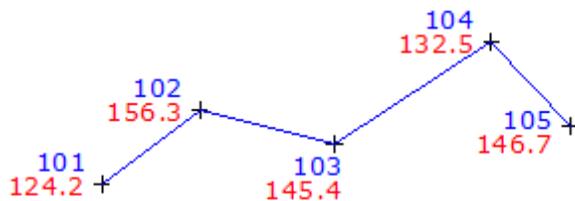
**To create a line by specifying a range of point numbers**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point # Range ⊕ .

2 Enter the number of the point at which you want to start the line.

3 Enter a hyphen (-).

4 Enter the number of the point at which you want to end the line segment and press Enter.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point # Range ⊕

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Point # Range

Command Line

Line, 'PN

# Creating Lines Through Specified Point Objects

The Line By Point Object command runs the AutoCAD Line command combined with the 'PO transparent command (page 1591) so you can select point objects through which to draw the line.

To select the points, you can select any part of the point objects, such as the marker or the label.



**To create a line by specifying point objects**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point Object .

2 Select the start point in the drawing.

3 Select the next point in the drawing.

4 Continue to select points to define additional line segments. Or press Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point Object 

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Point Object

Command Line

Line, 'PO

# Creating Lines by Specifying Point Names

The Line By Point Name command runs the AutoCAD Line command combined with the 'PA transparent command (page 1592) so you can specify the names of points through which to draw the line.

This command allows you to draw a line between points by specifying their names instead of their point numbers.
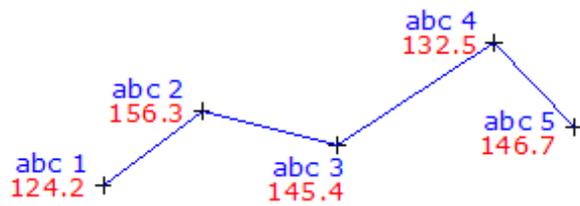
**To create a line by specifying point names**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point Name 🔣 .

2 Enter the name of the start point.

3 Enter the name of the next point.

4 Continue to enter point names to define additional line segments. Or press Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Point Name 🔣

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Point Name

Command Line

Line, 'PA

# Creating Lines by Specifying Northing/Easting Coordinates

The Line By Northing/Easting command runs the AutoCAD Line command combined with the 'NE transparent command (page 1588) so you can specify northing and easting coordinates of the line segments.

The order of the prompts is controlled by the Transparent Command Ambient drawing settings.
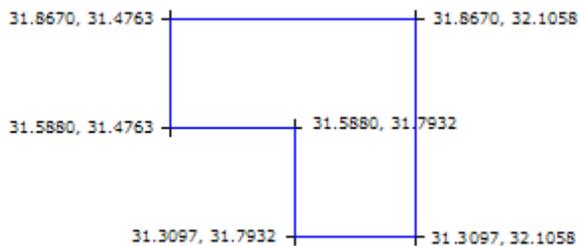
**To create a line by specifying northing and easting**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Northing/Easting .

**2** Enter a northing value.

**3** Enter an easting value.

> **NOTE** The order in which you are prompted for northing and easting is controlled by the Transparent Command setting Prompt For Easting Then Northing, located on the Ambient Settings tab of the Drawing Settings dialog box. For more information, see Specifying Ambient Settings (page 68).

**4** Continue to enter northing and easting values to define the line segments. Or press ESC to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Northing/Easting

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Northing/Easting

Command Line

Line, 'NE

# Creating Lines by Specifying Grid Northing/Easting Coordinates

The Line By Grid Northing/Grid Easting command runs the AutoCAD Line command combined with the 'GN transparent command (page 1589) so you can specify grid northing and easting coordinates of the line segments.

A coordinate system must be specified in the drawing. The order of the prompts is controlled by the Transparent Command Ambient drawing settings.
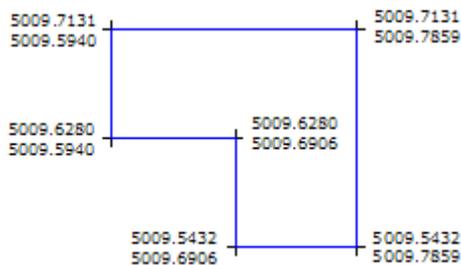


**To create a line by specifying grid northing and easting**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Grid Northing/Grid Easting .

**2** Enter a grid northing value.

**3** Enter a grid easting value.

**NOTE** The order in which you are prompted for grid northing and grid easting is controlled by the Transparent Command setting Prompt For Easting Then Northing, located on the Ambient Settings tab of the Drawing Settings dialog box. For more information, see Specifying Ambient Settings (page 68).

**4** Continue to enter grid northing and easting values to define the line segments. Or press ESC to return to the Line command prompt where you can use additional options to define the line.

### Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Grid Northing/Grid Easting ⟳

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Grid Northing/Grid Easting

Command Line

Line, 'GN

# Creating Lines by Specifying Latitudes and Longitudes

The Line By Latitude/Longitude command runs the AutoCAD Line command combined with the 'LL transparent command (page 1590) so you can specify latitude and longitude coordinates of the line segments.

A coordinate system must be specified in the drawing. The order of the prompts is controlled by the Transparent Command Ambient drawing settings.
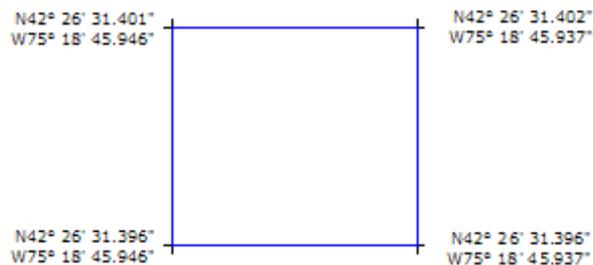


**To create a line by specifying latitude and longitude**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Latitude/Longitude ⟳ .

**2** Enter a latitude value.

**3** Enter a longitude value.

**NOTE** The order in which you are prompted for latitude and longitude is controlled by the Transparent Command setting Prompt For Longitude Then Latitude, located on the Ambient Settings tab of the Drawing Settings dialog box. For more information, see Specifying Ambient Settings (page 68).

**4** Continue to enter latitude and longitude values to define the line segments. Or press ESC to return to the Line command prompt where you can use additional options to define the line.

# Creating Lines by Specifying Bearings

The Line By Bearing command runs the AutoCAD Line command combined with the 'BD transparent command (page 1584) so you can specify a direction and distance for the line segment.

From the start point (1), specify the quadrant (2). Then specify the bearing (3) and distance (4).
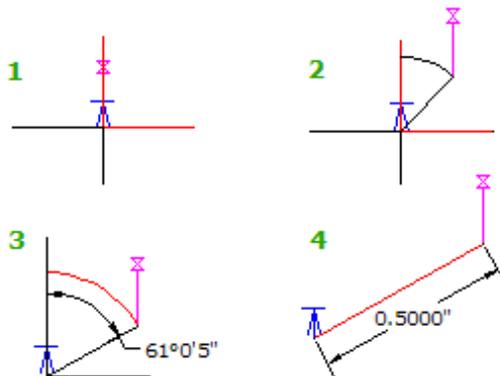


**To create a line by specifying bearings**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Bearing  .

2 Select a temporary start point for the angle measurement by clicking in the drawing or by using the .P, .N, or .G point filters (page 1599).

3 Specify a quadrant number by either clicking in the drawing or entering a value between 1 and 4.

4 Specify the bearing within the quadrant by clicking in the drawing or entering a bearing using the angular units for the drawing.

5 Specify the distance by either clicking in the drawing or entering a distance.

6 Continue to specify bearings and distances to define line segments. Or press Enter to return to the Line command prompt where you can use additional options to define the line.

**Quick Reference**

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Bearing

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Bearing

Command Line

Line, 'BD

# Creating Lines by Specifying Azimuths

The Line By Azimuth command runs the AutoCAD Line command combined with the 'ZD transparent command (page 1586) so you can specify a direction and distance for the line segment.

From the start point (1), specify the azimuth (2) and the distance (3).
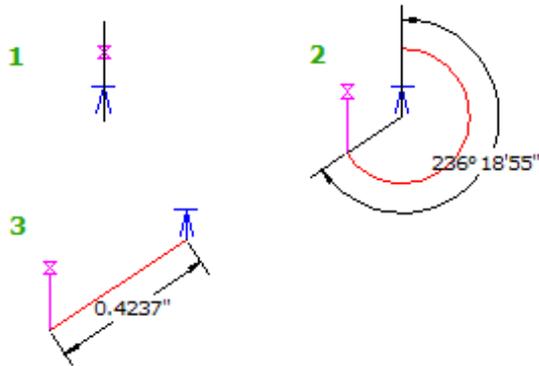


**To create a line by specifying azimuths**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Azimuth .

2 Select a temporary start point for the angle measurement by clicking in the drawing or by using the .P, .N, or .G point filters (page 1599).

3 Specify an azimuth by either clicking in the drawing or entering a value.

4 Specify the distance by either clicking in the drawing or entering a distance.

5 Continue to specify azimuths and distances to define line segments. Or press Enter to return to the Line command prompt where you can use additional options to define the line.

### Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Azimuth

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Azimuth

Command Line

Line, 'ZD

# Creating Lines by Specifying Angles

The Line By Angle command runs the AutoCAD Line command combined with the 'AD transparent command (page 1582) so you can specify an angle and a distance for the line segment.

The first sequence of prompts is for establishing a temporary reference line for measuring the angle. From the reference line (1-2), specify an angle (3), and distance (4), to specify a point (5).
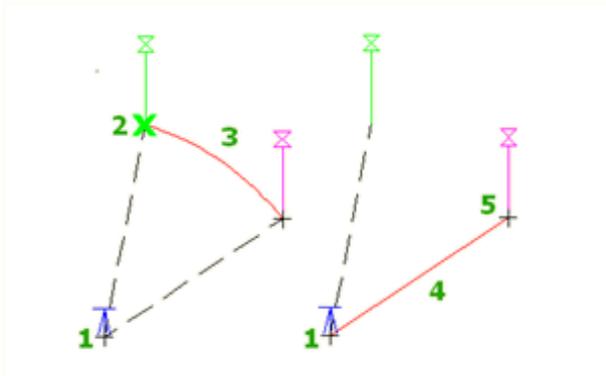


**To create a line by specifying angles**

1   Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Angle .

2   Specify a temporary reference line for the angle by doing one of the following:
    ■ Select a line.
    ■ Enter **Points**, or **P**, and then specify a start point and endpoint for the line.

3   Specify the angle by doing one of the following:
    ■ Click in the drawing to specify the angle.
    ■ Enter a positive or negative numeric value for the angle.
    ■ Enter **C** to switch the angle between counterclockwise and clockwise, and then specify the angle.

4   Specify the distance by either clicking in the drawing or entering a distance.

5   Specify an ending point.
    This concludes the specification of the temporary reference line. Subsequent prompts for angles and distances are then used to define the line segments.

6   Enter angles and distances to define line segments. You can then press Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

   Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Angle

Menu

   Lines/Curves menu ➤ Create Lines ➤ Line By Angle

Command Line

   Line, 'AD

# Creating Lines by Specifying Deflection Angles

The Line By Deflection command runs the AutoCAD Line command combined with the 'DD transparent command (page 1585) so you can specify a deflection angle and a distance.

The first sequence of prompts is for establishing a temporary reference line for measuring the angle.

From the reference line (1-2) and start point (2), specify a deflection angle (3), and distance (4) to specify a point (5).
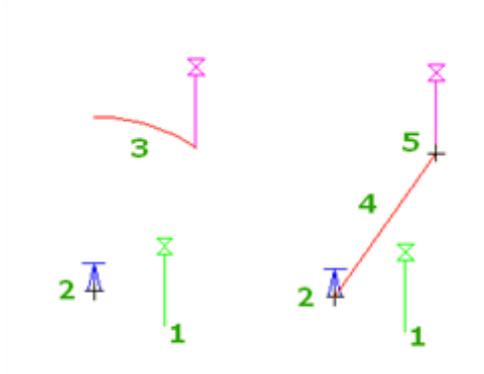


**To create a line by specifying deflection angles**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Deflection ⬕.

2 Specify a temporary reference line for the angle by doing one of the following:
   - Select a line.
   - Enter **Points**, or **P**, and then specify a start point and endpoint for the line.

3 Specify the angle by doing one of the following:
   - Click in the drawing to specify the angle.
   - Enter a positive or negative numeric value for the angle.
   - Enter **C** to switch the angle between counterclockwise and clockwise, and then specify the angle.

4 Specify the distance by either clicking in the drawing or entering a distance.

5 Specify an ending point.
   This concludes the specification of the temporary reference line. Subsequent prompts for angles and distances are then used to define the line segments.

6 Enter angles and distances to define line segments. You can then press Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

   Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Deflection ⬕
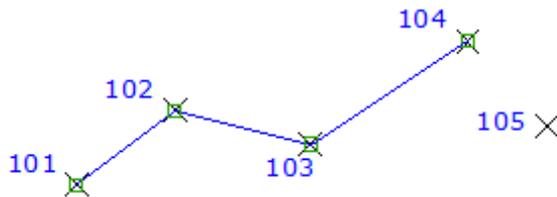
Menu

   Lines/Curves menu ➤ Create Lines ➤ Line By Deflection

Line, 'DD

# Creating Lines by Specifying Stations and Offsets

The Line By Station/Offset command runs the AutoCAD Line command combined with the 'SO transparent command (page 1592) so you can immediately specify a station and offset from an alignment.

An alignment must exist in the drawing to use this command.
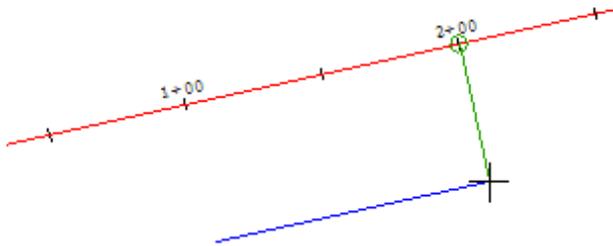


**To create a line by specifying station and offset from an alignment**

1  Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Station/Offset ⟋⸾ .

2  Select the alignment.

3  Specify a station by either entering a station value or clicking in the drawing.

4  Specify a distance by either entering an offset or clicking in the drawing.

5  Specify a station and offset for the next point in the line. Or press Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Station/Offset ⟋⸾

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Station/Offset

Command Line

Line, 'SO

# Creating Lines by Sideshot

The Line By Side Shot command runs the AutoCAD Line command combined with the 'SS transparent command (page 1587) so you can specify one or more point locations relative to a fixed point by specifying an angle (bearing, azimuth, angle, or deflection) and distance.

After defining the reference line, specify an angle and distance to the points that will define the line segment. From the reference line (1-2) and start point (2), specify points (3, 4, 5).
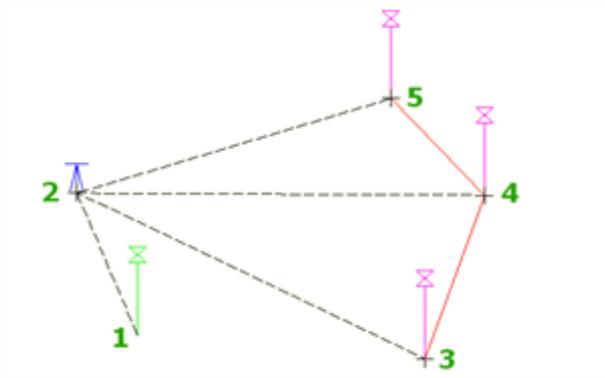


**To create a line by sideshot**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Side Shot .

2 Specify a temporary reference line by doing one of the following:

■ Select a line.

■ Enter **Points**, or **P**, and then specify a start point and endpoint for the line.

The following prompt is displayed:

`Specify angle or [Counter-clockwise/Bearing/Deflection/aZimuth]:`

3 Enter an angle by doing one of the following:

■ Enter an angle in angular units for the drawing.

■ Enter **Bearing**, or **B**, and then specify the angle by specifying a quadrant and a bearing.

■ Enter **Deflection**, or **D**, then specify a deflection angle.

■ Enter **aZimuth**, or **Z**, and then specify an azimuth.

■ Enter **C** to switch the direction of the angle between clockwise and counterclockwise.

4 Specify a distance by either clicking in the drawing or entering a distance.

5 Continue to specify angles and distances to define line segments. Or presss Enter to return to the Line command prompt where you can use additional options to define the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Side Shot 

Menu

Lines/Curves menu ➤ Create Lines ➤ Line By Side Shot

Command Line

Line, 'SS

# Extending Line Lengths

The Line Extension command extends an existing line by a specified distance or alters the line to match the total length specified.

The line is extended from the end of the existing line that is closest to the selection point.
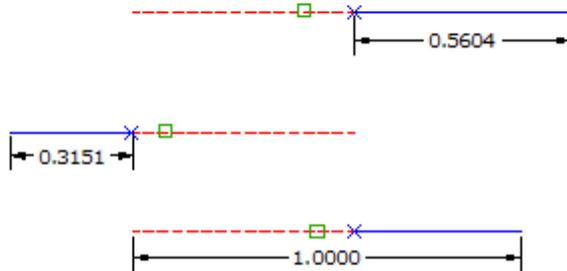


**To extend an existing line**

1 Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Extension .

2 Select the line to extend.

3 To define the length of the line, do one of the following:

■ Select two locations to specify the length that you want to add to the line.

■ Enter a positive distance to extend the line.

■ Enter a negative distance to shorten the line.

■ Enter **Total**, or **T**, and then enter the total length of the segment. You can either type the new total length, or select two locations to define the total length. This total length can be greater than (to lengthen the line), or smaller than (to shorten the line), the current length of the line.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line By Extension

Menu

Lines/Curves menu ➤ Create Lines ➤ Line Extension

Command Line

LineExtension

# Creating Lines From the End Of an Object

The Line From End Of Object command draws a tangent line from an existing object in the drawing.

The line is drawn from the end of the object that is closest to the selection point. Specify a distance by picking two points or entering a value. The line that is drawn from an end of an arc will be drawn tangent.
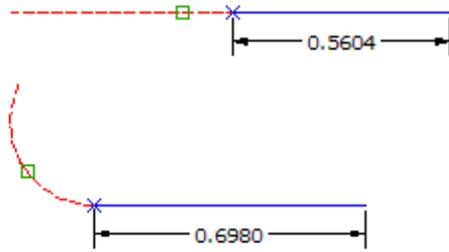
**To create a line from the end of an object**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line From End Of Object.

**2** Select the arc or line object to extend the line from.

**3** Specify a distance by either clicking in the drawing or entering a distance.

### Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line From End Of Object

Menu

Lines/Curves menu ➤ Create Lines ➤ Line From End Of Object

Command Line

LineFromEndOfObject

# Creating Lines Tangent to a Specified Point

The Line Tangent From Point command draws a tangent line from a selected point on an existing object in the drawing.

Select the location of the tangent point and then specify a distance by picking two points or entering a value.
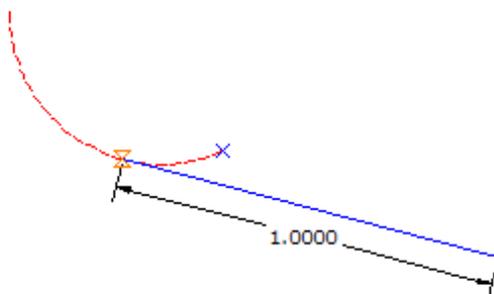
**To create a line tangent to a specified point**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Tangent From Point ⟍.

**2** Select the arc or line object to extend the line from.

**3** Specify the point of tangency on the object.

**4** Specify a distance by either clicking in the drawing or entering a distance.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Tangent From Point ⟍

Menu

Lines/Curves menu ➤ Create Lines ➤ Line Tangent From Point

Command Line

LineTangent

# Creating Perpendicular Lines

The Line Perpendicular From Point command draws a perpendicular or radial line from a selected point on an existing object in the drawing.

Select the location of the perpendicular point and then specify a distance by picking two points or entering a value.
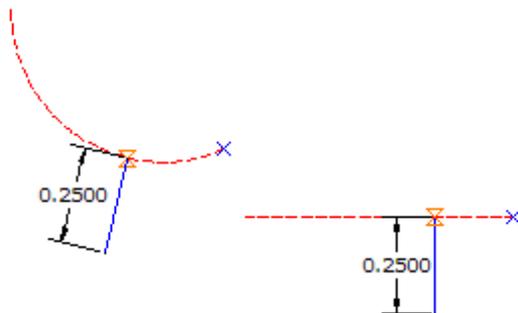


**To create a perpendicular or radial line**

**1** Click Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Perpendicular From Point ⟂.

**2** Select the arc or line object to extend the line from.

**3** Specify the point on the object where the line will extend from.

**4** Specify a distance by either clicking in the drawing or entering a distance.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Line drop-down ➤ Create Line Perpendicular From Point ⊥

Menu

Lines/Curves menu ➤ Create Lines ➤ Line Perpendicular From Point

Command Line

LinePerpendicular

# Creating Curves

Use the curve commands to draft curve geometry prior to defining it as alignments or parcels.

**The following illustration shows the curve parameters used by AutoCAD Civil.**
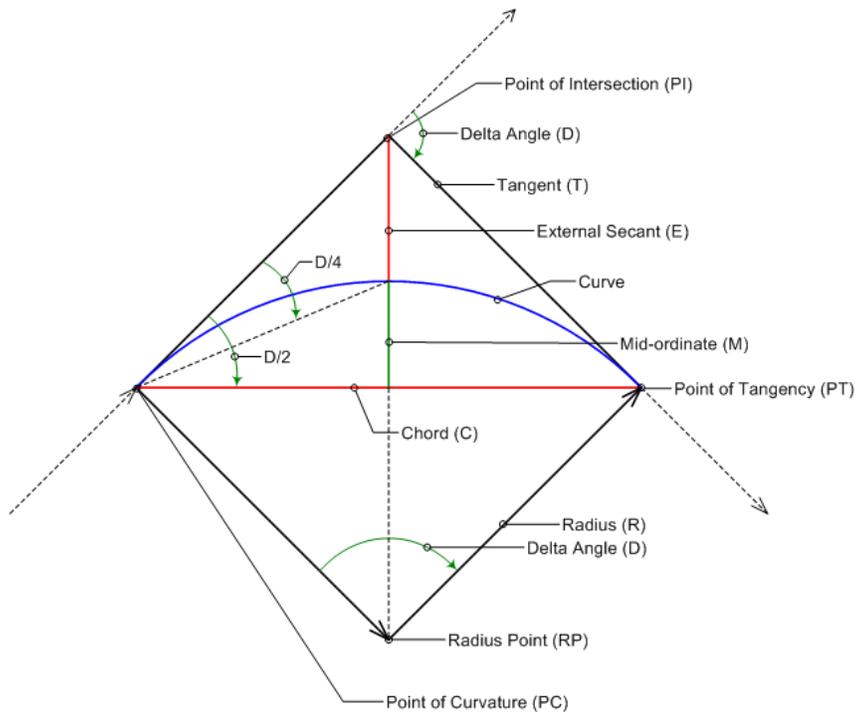
**The following illustration shows the degree of curve definition for arcs and chords.**

**Degree of Curve Arc Definition**



- Da
- 100 unit arc length
- Da
- R

**Degree of Curve Chord Definition**



- Dc
- 100 unit chord length
- Dc
- R
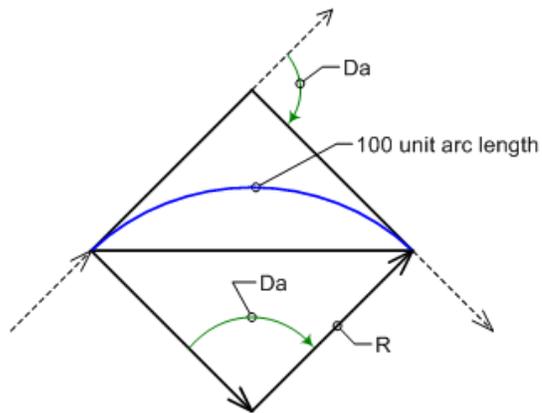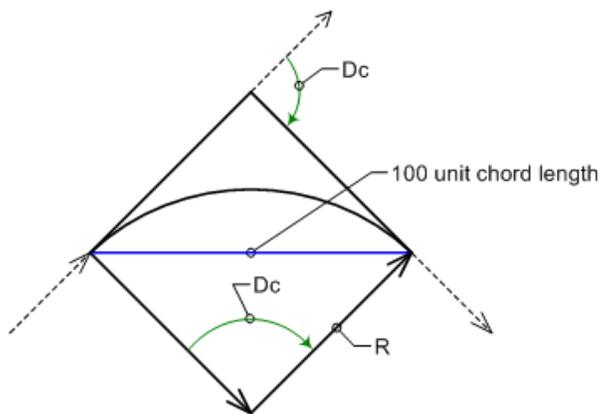
# Creating Curves Between Two Lines

The Curve Between Two Lines command creates a curve between two existing lines.

The selected lines are trimmed to the resulting point of curvature (PC) and point of tangency (PT).



You can use the Curve Calculator (page 593) to determine the values required for defining the curve.

**To create a curve between existing lines**

**1** Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curves Between Two Lines ⌒.

**2** Select the first tangent.

**3** Select the second tangent.
The following prompt is displayed:

```
Select entry [Tangent/External/Degree/Chord/Length/Mid-ordinate/miN-dist/Radius] <Radius>:
```

**4** Enter one of the following options to define the curve:

■ Enter **Length**, or **L**, and then enter the length or pick the distance in the drawing.

■ Enter **Tangent**, or **T**, and then enter the tangent length or pick the distance in the drawing.

■ Enter **External**, or **E**, and then enter the secant length or pick the distance in the drawing.

■ Enter **Degree**, or **D**, to define the curve by degree of curve.
The following prompt is displayed:

```
Enter degree of curve (Arc), or [Chord]:
```

Enter the degree of curve in DD.MMSS format using the Arc or Chord option.

Use the Chord option if the curve is a railway curve. For a railway curve, the degree of curve is the angle at the center of a circular curve subtended by a chord of 100 units.

Use the Arc option if the curve is a roadway curve. For a roadway curve, the degree of curve is the central angle subtended by a circular arc of 100 units.

■ Enter **Chord**, or **C**, and then enter the chord length or pick the distance in the drawing.

■ Enter **Mid-ordinate**, or **M**, and then enter the distance or pick the distance in the drawing.

■ Enter **miN-dist**, or **N**, and then select the end of an existing curve. Then enter the minimum distance between the end of the new curve and the point selected on the existing curve.

---

**NOTE** The command sets the Object Snap to END for this prompt. The point you select does not have to be on an existing curve; it can be any point along the adjacent tangent.

---

■ Enter **Radius**, or **R**, and then and then enter the radius or pick the distance in the drawing.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curves Between Two Lines ⌒

Menu

Lines/Curves menu ➤ Create Curves ➤ Curve Between Two Lines

Command Line

CurveBetweenTwoLines

# Creating Curves On Two Lines

The Curve On Two Lines command creates a curve between two existing lines.

The selected lines are not trimmed to the resulting point of curvature (PC) and point of tangency (PT).

You can use the Curve Calculator (page 593) to determine the values required for defining the curve.

**To create a curve between existing lines, without trimming the lines**

1  Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curve On Two Lines 𝐴 .

2  Select the first tangent.

3  Select the second tangent.
   The following prompt is displayed:

   ```
   Select entry [Tangent/External/Degree/Chord/Length/Mid-ordinate/miN-dist/Radius] <Radius>:
   ```

4  Enter one of the following options to define the curve:

   ■  Enter **Length**, or **L**, and then enter the length or pick the distance in the drawing.

   ■  Enter **Tangent**, or **T**, and then enter the tangent length or pick the distance in the drawing.

   ■  Enter **External**, or **E**, and then enter the secant length or pick the distance in the drawing.

   ■  Enter **Degree**, or **D**, to define the curve by degree of curve.
      The following prompt is displayed:

      ```
      Enter degree of curve (Arc), or [Chord]:
      ```

      Enter the degree of curve in DD.MMSS format using the Arc or Chord option.

      Use the Chord option if the curve is a railway curve. For a railway curve, the degree of curve is the angle at the center of a circular curve subtended by a chord of 100 units.

      Use the Arc option if the curve is a roadway curve. For a roadway curve, the degree of curve is the central angle subtended by a circular arc of 100 units.

   ■  Enter **Chord**, or **C**, and then enter the chord length or pick the distance in the drawing.

   ■  Enter **Mid-ordinate**, or **M**, and then enter the distance or pick the distance in the drawing.

   ■  Enter **miN-dist**, or **N**, and then select the end of an existing curve. Then enter the minimum distance between the end of the new curve and the point selected on the existing curve.

      **NOTE** The command sets the Object Snap to END for this prompt. The point you select does not have to be on an existing curve; it can be any point along the adjacent tangent.

   ■  Enter **Radius**, or **R**, and then and then enter the radius or pick the distance in the drawing.

**Quick Reference**

# Creating Curves Through a Specified Point

The Curve Through Point command creates a tangent arc that passes through a selected point between two line objects.

After selecting the two lines (1-2), select the pass-through point (3). The line objects are trimmed to the point of curvature (PC) and the point of tangency (PT).



**To create a curve between existing lines through a specified point**

1   Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curve Through Point ⌒.

2   Select the first tangent.

3   Select the second tangent.

4   Select the pass-through point.

**Quick Reference**

# Creating Multiple Curves

The Multiple Curves command creates up to ten adjoined curves between two lines.

You can use this command as an alternative to designing alignment curves with spiral sections because the curves can have different lengths and radii.
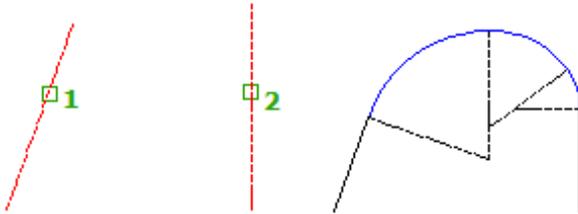


**To create multiple, adjoined curves**

1  Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Multiple Curves.

2  Select the first tangent.

3  Select the second tangent.

4  Specify the number of curves to insert between the two tangents. You can specify a maximum of 10.

5  Specify which curve will have a floating length. One curve in the set must have a floating length that is calculated by the lengths and radii of the other curves. This is usually a middle curve.

6  Enter the length and radius for all but the floating curve, and enter the radius of the floating curve.

If the results cannot fit between the two selected line objects, the following message is displayed:

```
Curves cannot fit between the tangents chosen.
Press any key to continue.
```

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Multiple Curves

Menu

Lines/Curves menu ➤ Create Curves ➤ Multiple Curves

Command Line

MultipleCurves

# Creating Curves From the End Of an Object

The Curve From End Of Object command creates a tangent arc from the end of a selected line or arc.

The curve is drawn from the end of the object that is closest to the object selection point.
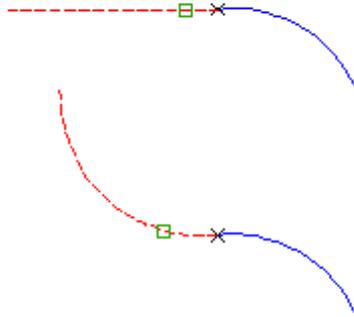
**To create a curve from the end of an object**

1. Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curve From End Of Object .

2. Select the line or arc nearest the end to which the new tangent arc is to be attached.

3. Specify one of the following types of entries to use:
   - **Point**: Enter **P** and then specify the end of the chord. The curve is then drawn.

   - **Radius**: Enter **R** and then do one of the following:
     - Enter the radius at the prompt.

     - Enter **A** and enter a degree of curve.

     - Enter **C** and enter a degree of chord.

   **NOTE** When drawing a curve, specifying a positive radius or degree of curve draws the curve clockwise or to the right of the starting angle, whereas a negative radius or degree of curve draws the type curve counterclockwise or to the left.

   After you enter the radius or degree of curve, the following prompt is displayed:

   ```
   Select entry [Tangent/Chord/Delta/Length/External/Mid-ordinate] <Length>:
   ```

4. Do one of the following:
   - Enter **Tangent**, or **T**, and then enter the tangent length, or pick the distance in the drawing.

   - Enter **Chord**, or **C** , and then enter the chord length, or pick the distance in the drawing.

   - Enter **Delta**, or **D**, and then enter the delta angle, or pick the distance in the drawing.

   - Enter **Length**, or **L**, and then enter the curve length, or pick the distance in the drawing.

   - Enter **External**, or **E**, and then enter the external distance, or pick the distance in the drawing.

   - Enter **Mid-ordinate**, or **M**, and then enter the mid-ordinate distance, or pick the distance in the drawing.

## Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Curve From End Of Object

Menu

Lines/Curves menu ➤ Create Curves ➤ Curve From End Of Object

Command Line

CurveFromEndOfObject

# Creating Reverse or Compound Curves

The Reverse or Compound command draws a reverse or compound curve from the endpoint of an existing curve.

A compound curve is a curve consisting of two or more arcs of different radii curving in the same direction and having a common tangent or transition curve at their point of junction. A reverse curve is an S-shaped curve.
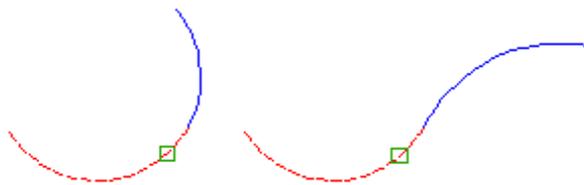
**To create a reverse or compound curve**

1 Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Reverse Or Compound Curve 〰.

2 Select the arc object nearest the end to which the new compound or reverse curve is to be attached.

3 Specify whether to create a Reverse or Compound curve.

4 Do one of the following:

■ Enter the radius at the prompt.

■ Enter **A** and enter a degree of curve.

■ Enter **C** and enter a degree of chord.

After you enter the radius or degree of curve, the following prompt is displayed:

```
Select entry [Tangent/Chord/Delta/Length/External/Mid-ordinate] <Length>:
```

5 Do one of the following:

■ Enter **Tangent**, or **T**, and then enter the tangent length, or pick the distance in the drawing.

■ Enter **Chord**, or **C** , and then enter the chord length, or pick the distance in the drawing.

■ Enter **Length**, or **L**, and then enter the curve length, or pick the distance in the drawing.

■ Enter **Delta**, or **D**, and then enter the delta angle, or pick the distance in the drawing.

■ Enter **External**, or **E**, and then enter the external distance, or pick the distance in the drawing.

■ Enter **Mid-ordinate**, or **M**, and then enter the mid-ordinate distance, or pick the distance in the drawing.

### Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Curves drop-down ➤ Create Reverse Or Compound Curve ⤳

Menu

Lines/Curves menu ➤ Create Curves ➤ Reverse Or Compound Curve

Command Line

ReverseOrCompound

# Creating Entities by Best Fit

Use the Best Fit Entities commands to use a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen as a basis to create AutoCAD lines, arcs, and parabolas.

The best fit commands are useful when you have to resurvey existing infrastructure during rehabilitation projects. In these cases, survey data may not match the original design. This requires you to edit the lines and curves in your design to best represent the actual survey data.

The best fit commands use the least square regression model to create lines, arcs, and parabolas from AutoCAD Civil points, AutoCAD points, clicks on screen, or existing entities such as lines, arcs, polylines, feature lines, etc. The line or curve that best fits through these objects represents the minimum deviation from the original design.

**NOTE** When using the best fit commands for profile design, use the Create Parabola command for vertical curves. See Vertical Curve Design (page 1068) for more information.

# Creating Lines by Best Fit

Use the create line by best fit command to use a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen as a basis to create an AutoCAD line.

The best-fitting line through the points is calculated. You can then use a dialog box to exclude any of the specified points or to select a pass-through point.

**To create an AutoCAD line by best fit from AutoCAD Civil points**

1   If you are creating a best fit line for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

2   Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Line.

3   In the Line By Best Fit Dialog Box (page 1989), select From COGO Points.

4   Select two or more points. Enter G to select a point group or N to enter points by number.

    As you select points in the drawing window, a white X marks each regression point and a temporary, dashed line is displayed in real time.

5   Press Enter to complete the command.

6   In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

    As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

    **NOTE** When the AutoCAD line is created, it will not retain any link to the regression points.

7   Create the AutoCAD line:

    ■   Click  to create the line and keep the regression data. You may then use the data as a basis to create more lines.

    ■   Click  to create the line, clear the regression data, and close the Regression Data vista.

8   If desired, convert the AutoCAD line to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD line by best fit from AutoCAD points**

1   If you are creating a best fit line for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

2   Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Line.

3   In the Line By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

4   Select two or more AutoCAD points.

    As you select points in the drawing window, a white X marks each regression point and a temporary, dashed line is displayed in real time.

5   Press Enter to complete the command.

6   In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

    As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

    **NOTE** When the AutoCAD line is created, it will not retain any link to the regression points.

**7** Create the AutoCAD line:

- ■ Click ![save icon] to create the line and keep the regression data. You may then use the data as a basis to create more lines.

- ■ Click ![checkmark icon] to create the line, clear the regression data, and close the Regression Data vista.

**8** If desired, convert the AutoCAD line to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD line by best fit from existing entities**

**1** If you are creating a best fit line for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Line ![icon].

**3** In the Line By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**4** Select one or more of the entities listed at the command line.

---
**NOTE** You may select several types of entities listed at the command line.

---

As you select entities in the drawing window, a white X marks each regression point and a temporary, dashed line is displayed in real time.

**5** If you selected a profile object, specify the starting and ending station on the Specify Station Range Dialog Box (page 1992).

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

---
**NOTE** When the AutoCAD line is created, it will not retain any link to the regression points.

---

**8** Create the AutoCAD line:

- ■ Click ![save icon] to create the line and keep the regression data. You may then use the data as a basis to create more lines.

- ■ Click ![checkmark icon] to create the line, clear the regression data, and close the Regression Data vista.

**9** If desired, convert the AutoCAD line to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD line by best fit by clicking on screen**

**1** If you are creating a best fit line for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Line ![icon].

**3** In the Line By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**4** Select a starting point and at least one other point.

> **NOTE** You may use OSNAP or transparent commands to select points.

As you select points in the drawing window, a white X marks each regression point and a temporary, dashed line is displayed in real time.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

> **NOTE** When the AutoCAD line is created, it will not retain any link to the regression points.

**7** Create the AutoCAD line:

- Click ⊟ to create the line and keep the regression data. You may then use the data as a basis to create more lines.

- Click ✔ to create the line, clear the regression data, and close the Regression Data vista.

**8** If desired, convert the AutoCAD line to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

### Quick Reference

Ribbon

Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Line ⁒⁒

Menu

Lines/Curves menu ➤ Create Curves ➤ Curve Between Two Lines

Command Line

CreateLineByBestFit

Dialog Box

Line By Best Fit (page 1989)
Specify Station Range (page 1992)
Regression Data Vista (page 1990)

# Creating Arcs by Best Fit

Use the create arc by best fit command to use a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen as a basis to create an AutoCAD arc.

The best-fitting curve through the points is calculated. You can then use a dialog box to exclude any of the specified points or to select one or two pass-through points.

**NOTE** When using the best fit commands for profile design, use the Create Parabola command for vertical curves. See Vertical Curve Design (page 1068) for more information.

**To create an AutoCAD arc by best fit from AutoCAD Civil points**

1   If you are creating a best fit arc for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

2   Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Arc ⬚⁄⬚ .

3   In the Arc By Best Fit Dialog Box (page 1989), select From COGO Points.

4   Select three or more points. Enter G to select a point group or N to enter points by number.

As you select points in the drawing window, a white X marks each regression point and a temporary, dashed arc is displayed in real time.

**NOTE** You must select at least three non-collinear regression points.

5   Press Enter to complete the command.

6   In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

**NOTE** When the AutoCAD arc is created, it will not retain any link to the regression points.

7   Create the AutoCAD arc:

   ■   Click 🖫 to create the arc and keep the regression data. You may then use the data as a basis to create more arcs.

   ■   Click ✔ to create the arc, clear the regression data, and close the Regression Data vista.

8   If desired, convert the AutoCAD arc to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD arc by best fit from AutoCAD points**

1   If you are creating a best fit arc for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Arc ⬚⌇ .

**3** In the Arc By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

**4** Select three or more AutoCAD points.

As you select points in the drawing window, a white X marks each regression point and a temporary, dashed arc is displayed in real time.

> **NOTE** You must select at least three non-collinear regression points.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

> **NOTE** When the AutoCAD arc is created, it will not retain any link to the regression points.

**7** Create the AutoCAD arc:

■ Click 💾 to create the arc and keep the regression data. You may then use the data as a basis to create more arcs.

■ Clickk ✔ to create the arc, clear the regression data, and close the Regression Data vista.

**8** If desired, convert the AutoCAD arc to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD arc by best fit from existing entities**

**1** If you are creating a best fit arc for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Arc ⬚⌇ .

**3** In the Arc By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**4** Select one or more of the entities listed at the command line.

> **NOTE** You may select several types of entities listed at the command line.

As you select entities in the drawing window, a white X marks each regression point and a temporary, dashed arc is displayed in real time.

> **NOTE** You must select at least three non-collinear regression points.

**5** If you selected a profile object, specify the starting and ending station on the Specify Station Range Dialog Box (page 1992).

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

> **NOTE** When the AutoCAD arc is created, it will not retain any link to the regression points.

**8** Create the AutoCAD arc:

- ■ Click ![icon] to create the arc and keep the regression data. You may then use the data as a basis to create more arcs.

- ■ Click ![icon] to create the arc, clear the regression data, and close the Regression Data vista.

**9** If desired, convert the AutoCAD arc to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD arc by best fit by clicking on screen**

**1** If you are creating a best fit arc for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Arc ![icon].

**3** In the Arc By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**4** Select a starting point and at least two other points.

> **NOTE** You may use OSNAP or transparent commands to select points.

As you select points in the drawing window, a white X marks each regression point and a temporary, dashed arc is displayed in real time.

> **NOTE** You must select at least three non-collinear regression points.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

> **NOTE** When the AutoCAD arc is created, it will not retain any link to the regression points.

**7** Create the AutoCAD arc:

- ■ Click ![icon] to create the arc and keep the regression data. You may then use the data as a basis to create more arcs.

- ■ Click ![icon] to create the arc, clear the regression data, and close the Regression Data vista.

**8** If desired, convert the AutoCAD arc to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**Quick Reference**

Ribbon

Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Arc

Menu

Lines/Curves menu ➤ Create Best Fit Entities ➤ Create Arc

Command Line

CreateArcByBestFit

Dialog Box

Arc By Best Fit (page 1989)
Specify Station Range (page 1992)
Regression Data Vista (page 1990)

# Creating Parabolas by Best Fit

Use the create parabola by best fit command to use a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen as a basis to create an AutoCAD parabola.

The best-fitting parabola through the points is calculated. You can then use a dialog box to exclude any of the specified points or to select one or two pass-through points.
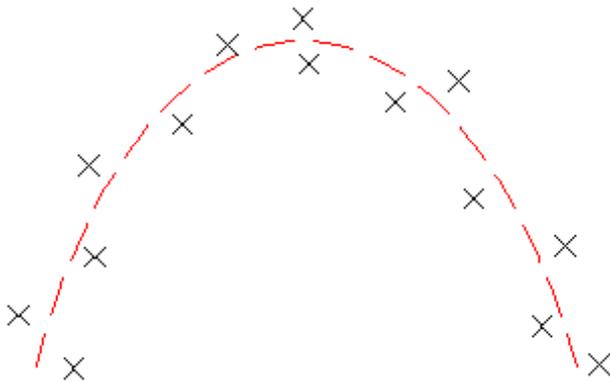


**NOTE** When using the best fit commands for profile design, use the Create Parabola command for vertical curves. See Vertical Curve Design (page 1068) for more information.

**To create an AutoCAD parabola by best fit from existing entities**

1   If you are creating a best fit parabola for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

2   Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Parabola.

3   In the Parabola By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

4   Select one or more of the entities listed at the command line.

**NOTE** You may select several types of entities listed at the command line.

As you select entities in the drawing window, a white X marks each regression point and a temporary, dashed parabola is displayed in real time.

**NOTE** You must select at least three non-collinear regression points.

**5** If you selected a profile object, specify the starting and ending station on the Specify Station Range Dialog Box (page 1992).

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

**NOTE** When the AutoCAD parabola is created, it will not retain any link to the regression points.

**8** Create the AutoCAD parabola:

- ■ Clik ![save icon] to create the parabola and keep the regression data. You may then use the data as a basis to create more parabolas.

- ■ Click ![checkmark icon] to create the parabola, clear the regression data, and close the Regression Data vista.

**9** If desired, convert the AutoCAD parabola to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

**To create an AutoCAD parabola by best fit by clicking on screen**

**1** If you are creating a best fit parabola for a profile, set the profile view style vertical exaggeration to 1. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Parabola ![parabola icon] .

**3** In the Parabola By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**4** Select a starting point and at least two other points.

**NOTE** You may use OSNAP or transparent commands to select points.

As you select points in the drawing window, a white X marks each regression point and a temporary, dashed parabola is displayed in real time.

**NOTE** You must select at least three non-collinear regression points.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window will be highlighted in red.

**NOTE** When the AutoCAD parabola is created, it will not retain any link to the regression points.

**7** Create the AutoCAD parabola:

- ■ Clik  to create the parabola and keep the regression data. You may then use the data as a basis to create more parabolas.

- ■ Click  to create the parabola, clear the regression data, and close the Regression Data vista.

**8** If desired, convert the AutoCAD parabola to an alignment or profile entity. See Converting an AutoCAD Line or Arc to an Alignment Sub-Entity (page 990) or Converting AutoCAD Entities to Profile Sub-entities (page 1096) for more information.

### Quick Reference

Ribbon

Click Home tab ➤ Draw panel ➤ Best Fit drop-down ➤ Create Best Fit Parabola 

Menu

Lines/Curves menu ➤ Create Curves ➤ Curve Between Two Lines

Command Line

CreateParabolaByBestFit

Dialog Box

Parabola By Best Fit (page 1989)
Specify Station Range (page 1992)
Regression Data Vista (page 1990)

# Attaching Multiple Lines or Curves to Existing Objects

The Attach Multiple command draws multiple lines and curves from the end of an existing line or arc.

The lines or curves are drawn from the end of the object closest to the selection point.
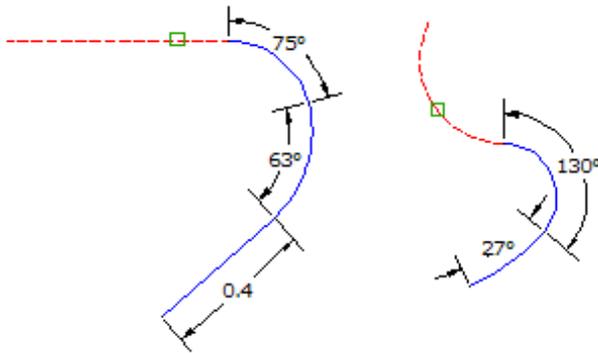


**To attach multiple lines**

**1** Click Home tab ➤ Draw panel  Attach Multiple.

**2** Select a line or arc.

**3** Enter **Line**, or **L**, to attach a line.

**4** Enter the length of the new line section.

The new line is drawn and the following prompt is displayed:

```
Enter an attachment option [Line/Arc/Undo]:
```

**5** Continue to add line or arc segments or use the Undo option to undo the last segment.

**To attach multiple curves**

**1** Click Home tab ➤ Draw panel ▼ Attach Multiple 🔲 .

**2** Select a line or arc.

**3** Enter **Arc**, or **A**, to attach an arc.

**4** Specify one of the following types of entries to use:

- **Point**: Enter **P** and then specify the end of the chord. The curve is then drawn.

- **Radius**: Enter **R** and then do one of the following:

  - Enter the radius at the prompt.

  - Enter **A** and enter a degree of curve.

  - Enter **C** and enter a degree of chord.

---

**NOTE** When drawing a curve, specifying a positive radius or degree of curve draws the curve clockwise or to the right of the starting angle, whereas a negative radius or degree of curve draws the type curve counterclockwise or to the left.

---

After you enter the radius or degree of curve, the following prompt is displayed:

```
Select entry [Tangent/Chord/Delta/Length/External/Mid-ordinate] <Length>:
```

**5** Do one of the following:

- Enter **Tangent**, or **T**, and then enter the tangent length, or pick the distance in the drawing.

- Enter **Chord**, or **C** , and then enter the chord length, or pick the distance in the drawing.

- Enter **Delta**, or **D**, and then enter the delta angle, or pick the distance in the drawing.

- Enter **Length**, or **L**, and then enter the curve length, or pick the distance in the drawing.

- Enter **External**, or **E**, and then enter the external distance, or pick the distance in the drawing.

- Enter **Mid-ordinate**, or **M**, and then enter the mid-ordinate distance, or pick the distance in the drawing.

The new line is drawn and the following prompt is displayed:

```
Enter an attachment option [Line/Arc/Undo]:
```

**6** Continue to add line or arc segments or use the Undo option to undo the last segment.

## Quick Reference

Ribbon

Click Home tab ➤ Draw panel ▼ Attach Multiple Entities 🔲

Menu

> Click Lines/Curvesmenu ➤ Attach Multiple Entities.

Command Line

> AttachMultiple

# Calculating Curve Parameters

The Curve Calculator determines curve parameters based on input.

You can keep the Curve Calculator open while you do other work and use the buttons to send the output of the curve calculations to the command line.

**NOTE** To change the units used in the Curve Calculator dialog box, use the Ambient Settings tab of the Drawing Settings dialog box. Under Angle, change the Format value to the unit you want to use. For more information, see To specify ambient settings for a drawing (page 68).

**The following illustration shows the curve parameters used by AutoCAD Civil.**
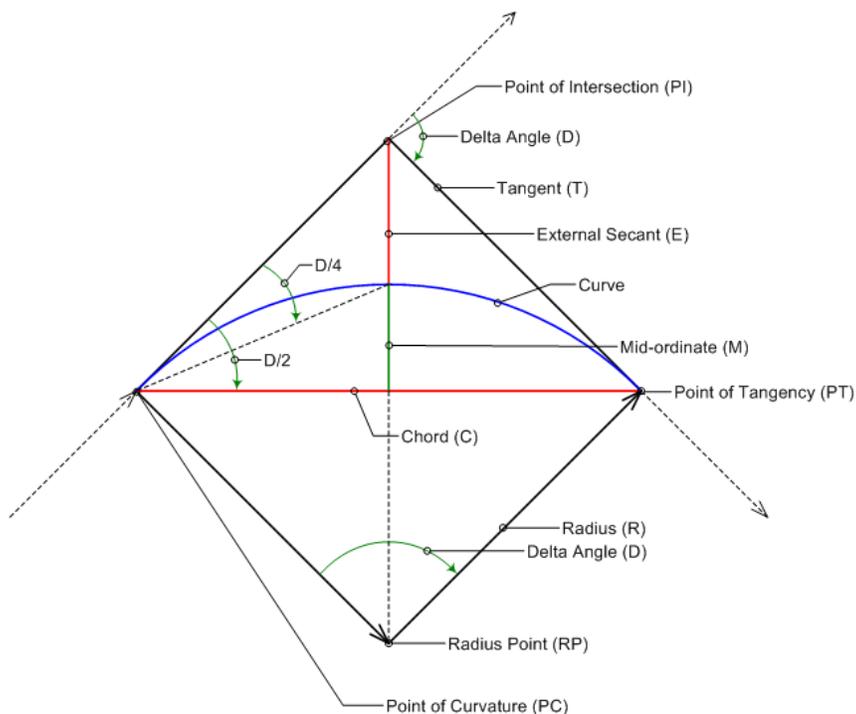


**To calculate curve parameters**

1   Click Home tab ➤ Draw panel ➤ Curves drop-down ➤ Curve Calculator ⬚ .

2   Specify whether the degree of curve is determined by arc or by chord by selecting an option from the Degree of Curve Definition drop-down list. The selected option affects how the Degree of Curve Property is calculated.

   ■ **Chord Definition**: Use this option if the curve is a railway curve. Using this option, the degree of curve is the angle at the center of a circular curve subtended by a chord of 100 units.

   ■ **Arc Definition**: Use this option if the curve is a roadway curve. Using this option, the degree of curve is the central angle subtended by a circular arc of 100 units.

**3** Specify a Fixed Property for the curve by selecting one of the following options:

- **Radius**: Specifies that the radius will be fixed.

- **Delta Angle**: Specifies that the delta angle will be fixed.

The program then holds as fixed either of the two parameters above while performing calculations.

**4** Specify one or more additional parameters for the curve.

- **Degree Of Curve**: Specifies the degree of curve.

- **Delta Angle**: Specifies the delta angle of the curve. This option is not adjustable if Delta Angle is specified as the Fixed Property.

- **Radius**: Specifies the radius of the curve. This option is not adjustable if Radius is specified as the Fixed Property.

- **Tangent Distance**: Specifies the tangent length of the curve.

- **Arc Distance**: Specifies the arc length of the curve.

- **Chord Distance**: Specifies the chord length of the curve.

- **External Distance**: Specifies the external secant length of the curve.

- **Mid-ordinate Distance**: Specifies the mid-ordinate length of the curve.

**5** Optionally you can also:

- Click  to send any of the values to the AutoCAD command line.

- Right-click and select Copy Value To Clipboard to copy the selected value to the clipboard.

- Right-click and select Copy To Clipboard to copy the entire contents of the Curve Calculator to the clipboard.

- Click  to select an arc in the drawing to display its values in the calculator. No edits you subsequently make in the calculator are updated to the arc in the drawing.

### Quick Reference

Ribbon

Home tab ➤ Draw panel ➤ Curves drop-down ➤ Curve Calculator 

Menu

Lines/Curves menu ➤ Curve Calculator

Command Line

CurveCalculator

Dialog Box

Curve Calculator (page 1992)

# Labeling Lines, Curves, and Polylines

You can label individual line and curve segments, one at a time, or multiple segments at once by using specified line and curve label styles.

# Labeling Individual Line and Curve Segments

You can label individual line and curve segments on lines, curves, polylines, and feature lines.

On the Toolspace Settings tab, in the General Label Style collection, set up Line and Curve label styles (page 1479) to use for the lines, curves and polylines.

On the Toolspace Settings tab, in the General Label Style collection, specify the default line label style (page 1476).

Use the Single Segment or Multiple Segment command to quickly label a line or curve using the default label style.

Use the Add Line/Curve Labels option to access the Add Labels dialog box (page 1970) if you need to create or modify a style before labeling a line or curve segment.

**To label individual line and curve segments**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Single Segment Line/Curve Label 🏷 .

2 Select the line segment to label.

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Line And Curve Labels 🏷 .

2 In the Add Labels dialog box (page 1970), under Feature, ensure that Line And Curve is selected.

3 Under Label Type, select Single Segment.

4 Specify the other options as required.

5 Click Add and then select the feature line or segment to label.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Single Segment Line/Curve Label 🏷

Menu

Lines/Curves menu ➤ Add Line/Curve Labels ➤ Single Segment

# Labeling Multiple Line and Curve Segments

You can label multiple line and curve segments on lines, curves, polylines, and feature lines.

**To label multiple line and curve segments**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Multiple Segment Line/Curve Labels 🏷 .

2 Select the lines and curve segments to label.

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Line And Curve Labels 🏷 .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Line And Curve is selected.

**3** Under Label Type, select Multiple Segment.

**4** Specify the other options as required.

**5** Click Add and then select the feature line or segment to label.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Multiple Segment Line/Curve Labels

🏷

Menu

Lines/Curves menu ➤ Add Line/Curve Labels ➤ Multiple Segment

# Adding Line/Curve Tables

Use line/curve tables to consolidate information about the line and arc segments in your drawing.

Tables provide an alternative to labels for information management. Tables keep the information organized and separate from the drawing, but cross-referenced by small labels known as tags.

When segment information is displayed in table rows, table tags uniquely identify the segments in the drawing and in the initial cells of table rows. For example, if a line segment is labeled L234, its corresponding table row is also labeled L234.

The line and curve table creation commands are available from the Labels & Tables panel on the Annotate tab.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. For general information about modifying tables, see Modifying Tables (page 1567).

# Renumbering Table Tags

Use the renumbering command to renumber table tags before creating tables.

In this release of AutoCAD Civil, the table tag renumbering command is available from the Tables contextual tab. It is also available from the Labels & Tables panel on contextual tabs for objects that support tags, such as Alignment and Parcel.

For more information, see Renumbering Label Tags (page 1544). The Table Tag setting process is now a drawing-wide setting. See Creating Label Tags (page 1544) for more information.

# Lines and Curves Command Reference

Quickly access Lines and Curves functionality from the command line.

| Command | Description |
|---|---|
| AttachMultiple | Attaches multiple lines or curves to existing entities (page 591) |
| CreateArcByBestFit | Creates an arc by best fit (page 585) |
| CreateLineByBestFit | Creates an AutoCAD line by best fit (page 582) |
| CreateParabolaByBestFit | Creates an AutoCAD parabola by best fit (page 589) |
| CurveBetweenTwoLines | Creates a curve between two existing lines (page 575) |
| CurveCalculator | Calculates curve parameters based on input. (page 593) |
| CurveFromEndOfObject | Creates a tangent arc from the end of a selected line or arc (page 579) |
| CurveOnTwoLines | Ceates a curve between two existing lines, but does not trim the selected lines (page 576) |
| CurveThroughPoint | Creates a tangent arc that passes through a selected point between two line objects (page 578) |
| LineExtension | Extends an existing line (page 571) |
| LineFromEndOfObject | Draws a tangent line from an existing object in the drawing (page 571) |
| LinePerpendicular | Draws a perpendicular or radial line from a selected point on an object (page 573) |
| LineTangent | Draws a tangent line from a selected point on an object (page 572) |
| MultipleCurves | Creates multiple,adjoined curves between two lines (page 579) |
| ReverseOrCompound | Draws a reverse or compound curve from the end-point of an existing curve (page 581) |

# Surfaces

# 19

Surfaces are basic building blocks in AutoCAD Civil. You can import surface information from LandXML, TIN (page 2485) (triangulated irregular network), and DEM (page 2472) (digital elevation model) files, and use points, point files, DEM data, existing AutoCAD objects, contours, breaklines, and boundaries to create surfaces.

Surfaces Workflow

## Understanding Surfaces

A surface is a three-dimensional geometric representation of an area of land, or, in the case of volume surfaces, is a difference or composite between two surface areas.

Surfaces are made up of triangles or grids, which are created when AutoCAD Civil connects the points that make up the surface data.

To use a surface in your drawing, you can create an empty surface and then add data to it. You can also import existing files containing surface information, such as LandXML, TIN (page 2485), or DEM (page 2472) files.

Points or contours are usually a primary part of the original surface information and are supplemented with breakline (page 2469) and boundaries (page 2469).

## Types of Surfaces

AutoCAD Civil supports several types of surfaces:

■   **TIN surfaces**. Formed by triangulating an arbitrary set of points.
For more information, see Creating a TIN Surface (page 601).

■   **Grid surfaces**. Formed from points that lie on a regular grid (for example, Digital Elevation Models (DEMs)).
For more information, see Creating a Grid Surface (page 603).

■   **TIN volume surfaces**. A composite surface created from a combination of points in a top (comparison) and base surface, also known as a differential surface.
For more information, see Creating a TIN Volume Surface (page 604).

- ▣ **Grid volume surfaces**. A differential surface based on user-specified top and bottom surfaces with points on a user-specified grid.
  For more information, see Creating a Grid Volume Surface (page 606).

## The Surface Object

Surfaces persist in an AutoCAD drawing as objects with the names AECC_TIN_SURFACE (TIN surface) or AECC_GRID_SURFACE (grid surface).

All AutoCAD Civil objects are aware of each other and react to changes. For example, a change in a surface results in multiple automatic updates in alignments, parcels, or other objects that reference that surface.

For overview information about AutoCAD Civil objects, see Understanding Objects and Styles (page 51).

## Surfaces Collection (Prospector Tab)

Use the Surfaces collection in the Prospector tree to access the surfaces in a drawing. As surface objects are created, they are displayed as named surfaces under the Surfaces collection.

Right-click the Surfaces collection to do the following:

- Create a new surface. (page 601)

- Import DEM data to create a surface. (page 608)

- Import TIN data to create a surface. (page 607)

- Turn on or off the surface preview (page 684). A check mark is displayed next to the Show Preview menu item when the preview option is selected.

- Export surface data to DEM. (page 714)

- Export the surface to LandXML format. (page 714)

- Refresh the view of the Surfaces collection in the Prospector tree.

If you have added a surface to the current drawing, you can expand the Surfaces collection to view the names of the surfaces or select surfaces to display a tabular list of the surfaces in the Prospector list view. For more information, see The Toolspace Item View (page 82).

Expand an individual surface name to display the surface components, including its masks (page 2477), watersheds (page 2487), and definition.

Expand the Definition collection to view and add data for the surface, such as breaklines, boundaries, and contours. The data items that are displayed depend on the surface type.

| For more information about... | See... |
| --- | --- |
| Masks | Masks (page 669) |
| Watersheds | Watersheds (page 675) |
| Definition | Understanding the Surface Definition (page 611) |

# Surface Collection (Settings Tab)

Use the Surface collection in the Settings tree to manage surface settings, surface styles, surface label styles, surface table styles, and surface command settings.

Right-click the Surface collection to do the following:

■ Edit the surface feature settings. (page 687)

■ Edit the surface label style defaults. (page 696)

■ Refresh the display of the settings tree.

Expand the Surface collection to display and edit the styles and command settings that are available for surfaces.

| For more information about... | See... |
|---|---|
| Surface styles | Surface Styles and Visualization (page 692) |
| Label styles | Surface Label Styles (page 696) |
| Table styles | Surface Table Styles (page 706) |
| Commands | Surfaces Command Reference (page 715)<br>Command Settings (page 687) |

# Creating Surfaces

You can create a surface that comprises a combination of points, breaklines, boundaries, and contours.

When you create a surface, the surface name is displayed in the ⬦ Surfaces collection in the Prospector tree so that you can perform other operations, such as adding data and editing the surface.

Initially, the surface may be empty and not be visible in the drawing. Once you add data to the surface, the surface becomes visible in the drawing in accordance to the display settings specified in the referenced surface style. See Surface Styles and Visualization (page 692).

# Creating a TIN Surface

A TIN (page 2485) surface comprises the triangles that form a triangulated irregular network.

TIN lines form the triangles that make up the surface triangulation. To create TIN lines, AutoCAD Civil connects the surface points that are closest together. The elevation of any point in the surface is defined by interpolating the elevations of the vertices of the triangles that the point lies in.

TIN surfaces are most useful for:

■ Mapping highly variable surfaces with irregularly distributed sample data representing the influence of streams, roads, and lakes.

■ Examining localized areas (large-scale maps)

TIN surfaces generally take longer to build and require more disk space than grid surfaces.

When AutoCAD Civil creates a TIN surface from point data, it computes the Delaunay triangulation (page 2472) of the points. With Delaunay triangulation, no point lies inside the circle determined by the vertices of any triangle.

Breakline data (from breaklines, contours, or boundaries) influences how the surface is triangulated. A breakline edge between the points causes the program to connect these points with a triangle edge in the TIN, even if doing so violates the Delaunay property.

A TIN surface with contour lines:
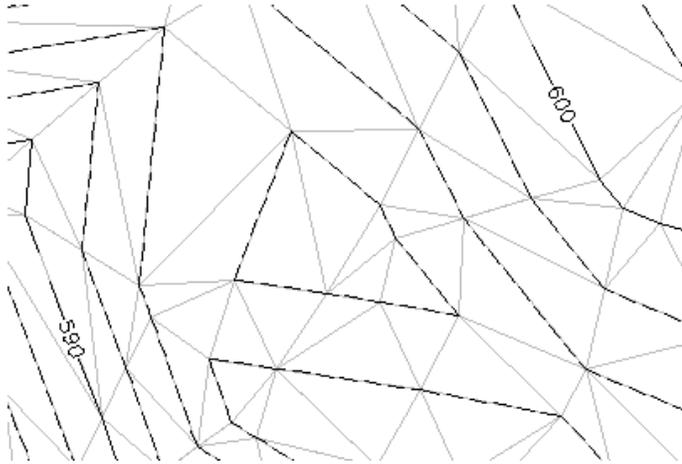


**To create a TIN surface**

1  Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface 🖼️ .
   In the Create Surface (page 2354) dialog box, in the Type list, select TIN Surface.

2  Click 🗐 to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

   **NOTE** If you do not select a layer, the surface is placed on the current layer.

3  In the properties grid, click the Value column for the Name property and enter a name for the surface.

   **NOTE** To name the surface, click its default name and enter a new name, or use the Name Template. For more information, see Name Template Dialog Box (page 1806).

4  To change the style for the surface, click the Style property in the properties grid and click ⬜ in the Value column.

   The Select Surface Style dialog box is displayed. For more information, see Surface Styles and Visualization (page 692).

5  To change the render material for the surface, click the Render Material property in the properties grid and click ⬜ in the Value column.

   The Select Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578)

6  Click OK to create the surface.

   The surface name is displayed under the Surfaces collection in the Prospector tree. For information about adding data to the surface, see Adding and Editing Surface Data (page 611).

**Quick Reference**

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface 

Menu

Surfaces menu ➤ Create Surface

Toolspace Shortcut Menu

Prospector tab: right-click Surfaces ➤ Create Surface

Command Line

CreateSurface

Dialog Box

Create Surface (page 2354)

# Creating a Grid Surface

A grid surface (page 2475) comprises points that lie on a regular grid.

You can create a grid surface or import it from a DEM (page 2472) file.

Use grid surfaces for:

■ Mapping more uniform surfaces with evenly distributed sample data

■ Examining large study areas (small-scale maps)

Grid surfaces generally load quickly and require less disk space than TIN surfaces.

**To create a grid surface**

1  Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface  .
   In the Create Surface (page 2354) dialog box, in the Type list, select Grid Surface.

2  Click  to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

   **NOTE** If you do not select a layer, the surface is placed on the current layer.

3  In the properties grid, click the Value column for the Name property and enter a name for the surface.

   **NOTE** To name the surface, click its default name and enter a new name, or use the name template. See Name Template Dialog Box (page 1806).

4  To change the style for the surface, click the Style property in the properties grid and click  in Value column.
   The Select Surface Style dialog box is displayed. See Surface Styles and Visualization (page 692).

5  To change the render material for the surface, click the Render Material property in the properties grid and click  in the Value column.
   The Select Render Material dialog box is displayed. See Applying Render Materials to Objects (page 1578)

**6** Set the Grid Parameters, such as spacing and orientation. Enter their values or click  to use the drawing area to derive these values.

**7** Click OK to create the surface.

After the surface is created, the surface name is displayed in the Surfaces collection in the Prospector tree, so that you can perform other operations, such as adding data and editing the surface.

For information about adding data to the surface, see Adding and Editing Surface Data (page 611).

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface

Menu

Surfaces menu ➤ Create Surface

Toolspace Shortcut Menu

Prospector tab: right-click Surfaces ➤ Create Surface

Command Line

CreateSurface

Dialog Box

Create Surface (page 2354)

## Creating a TIN Volume Surface

A TIN volume surface is a composite of points in a base surface (page 2468) and comparison surface (page 2470).

A TIN volume surface provides an exact difference between the base and comparison surfaces. Therefore, the Z-value of any point in the volume surface is precisely the difference between the Z of the comparison surface at that point and the base surface at that point. This is true whether the comparison and base surfaces are both grid surface, both TIN surfaces, or one of each.

A volume surface is a persistent surface object. Therefore, you can display cut and fill contours, cut and fill points, and add labels to it. You can view volume properties (cut, fill, net) of a volume surface by selecting Surface Properties. For more information, see Editing and Viewing the Surface Definition (page 681).

If you want only to query and obtain information about a surface volume or bounded volume (page 2469), use the Volumes and Bounded Volumes utilities.

**See also:**

■ Calculating Surface Volumes (page 690)

**To create a TIN volume surface**

**1** Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface .
In the Create Surface (page 2354), in the Type list, select TIN Volume Surface.

**2** Click to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

**NOTE** If you do not select a layer, the surface is placed on the current layer.

**3** In the properties grid, click the Value column for the Name property and enter a name for the surface.

**NOTE** To name the surface, click its default name and enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**4** To change the style for the surface, click the Style property in the properties grid and click ⋯ in the Value column.

The Select Surface Style dialog box is displayed. For more information, see Surface Styles and Visualization (page 692).

**5** To change the render material for the surface, click the Render Material property in the properties grid and click ⋯ in the Value column.

The Select Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578)

**6** Click the Base Surface property to select the base (bottom) surface. Enter the surface name or click ⋯ to open the Select Base Surface (page 1810)dialog box where you can select the surface in the list.

**NOTE** Select a surface in the drawing by clicking 🖳 and following the instructions at the command line.

**7** Click the Comparison Surface property to select the comparison (top) surface. Enter the surface name or click ⋯ to open the Select Comparison Surface dialog box where you can select the surface in the list.

**NOTE** Select a surface in the drawing by clicking 🖳 and following the instructions at the command line.

**8** Click OK to create the surface.

The surface name is displayed under the Surfaces collection in the Prospector tree. For information about adding data to the surface, see Adding and Editing Surface Data (page 611).

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface 🏠

Menu

Surfaces menu ➤ Create Surface

Toolspace Shortcut Menu

Prospector tab: Right-click Surfaces ➤ Create Surface

Command Line

CreateSurface

Dialog Box

Create Surface (page 2354)

# Creating a Grid Volume Surface

A grid volume surface is a difference grid surface based on user-specified base and comparison surfaces as well as grid spacing and orientation.
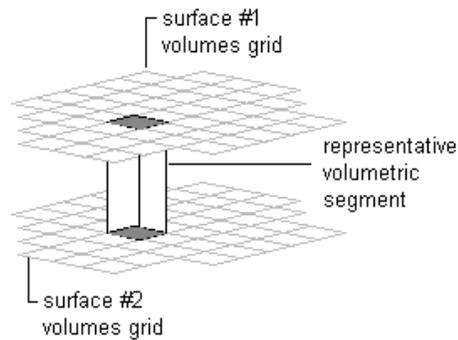


A grid volume surface is an approximation of the difference between the base and comparison surfaces. It is formed by making a grid from points whose Z values are the differences between the Z values of the comparison and base surfaces. Therefore, the grid volume surface gives the exact difference only at the grid points. This is true whether the comparison and base surfaces are both grid surfaces, both TIN surfaces, or one of each.

Grid volume surfaces enable you to generate a volume quickly, which is useful for iterative site design.

**NOTE** Grid spacing has a direct correlation as to how quickly grid volumes generate.

A volume surface is a persistent surface object. Therefore, you can display cut and fill contours, cut and fill points, and add labels to it. View a volume property (cut, fill, net) of a volume surface by selecting Surface Properties. For more information, see Editing and Viewing the Surface Definition (page 681).

If you want only to query and obtain information about a surface volume or a bounded volume (page 2469), use the Volumes and Bounded Volumes utilities.

**See also:**

■ Calculating Surface Volumes (page 690)

**To create a grid volume surface**

1 Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface ![icon] .
   In the Create Surface (page 2354) dialog box, in the Type list, select Grid Volume Surface.

2 Click ![icon] to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

   **NOTE** If you do not select a layer, the surface is placed on the current layer.

3 In the properties grid, click the Value column for the Name property and enter a name for the surface.

   **NOTE** To name the surface, click its default name and enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

4 To change the style for the surface, click the Style property in the properties grid and click ![icon] in the Value column.
   The Select Surface Style dialog box is displayed. For more information, see Surface Styles and Visualization (page 692).

**5** To change the render material for the surface, click the Render Material property in the properties grid and click ⬚ in the Value column.

The Select Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578)

**6** Click the Base Surface property to select the base (bottom) surface. You can enter the surface name or click ⬚ to open the Select Base Surface (page 1810) dialog box where you can select the surface in the list.

---

**NOTE** Select a surface in the drawing by clicking 🔲 and following the instructions at the command line.

---

**7** To set the grid parameters, such as spacing and orientation, enter their values into the Values column or click 🔲 to use the drawing area to derive these values.

---

**NOTE** The grid spacing parameters determine how precise a surface volume is. For example, small grid spacing creates a precise volume measurement (but may increase the loading time).

---

**8** Click the Comparison Surface property to select the comparison (top) surface. Enter the surface name or click ⬚ to open the Select Comparison Surface dialog box where you can select the surface in the list.

---

**NOTE** Select a surface in the drawing by clicking 🔲 and following the instructions at the command line.

---

**9** Click OK to create the surface.

The surface name is displayed under the Surfaces collection in the Prospector tree. For information about adding data to the surface, see Adding and Editing Surface Data (page 611).

### Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface 🖼️

Menu

Surfaces menu ➤ Create Surface

Toolspace Shortcut Menu

Prospector tab: Right-click Surfaces ➤ Create Surface

Command Line

CreateSurface

Dialog Box

Create Surface (page 2354)

# Creating a Surface From a TIN File

Use the Create Surface From TIN dialog box to create a surface by importing TIN (page 2485) format data.

Importing a TIN file creates a surface object on which you can perform all the standard data operations and editing. The import operation is displayed as the operation type Import Surface. See Understanding the Surface Definition (page 611).

You can use this command to import a TIN file from an Autodesk Land Desktop project. To import the TIN file, the accompanying PNT point file must exist in the same source directory.

**To import a surface from a TIN file**

1   Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From TIN  .

2   In the Create Surface From TIN dialog box, browse to the location of the TIN file and select it.

3   Click Open to add the surface to the current drawing.

    The surface name is displayed under the Surfaces collection in the Prospector tree. For information about adding data to the surface, see Adding and Editing Surface Data (page 611).

## Quick Reference

Ribbon

  Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From TIN 

Menu

  Surfaces menu ➤ Create Surface From TIN

Toolspace Shortcut Menu

  Prospector tab: right-click Surfaces  ➤  Create Surface from TIN

Command Line

  CreateSurfaceFromTIN

Dialog Box

  Create Surface From TIN (standard file selection dialog box)

# Creating a Grid Surface from a DEM

Use the Grid Surface From DEM dialog box to create a grid surface.

A DEM file comes in several formats. You can import either USGS DEM, ESRI Arc Grid, or GeoTIFF files.

**TIP** You can also import DEM files into an existing surface. For more information, see Adding DEM Files to a Surface (page 637).

For more information about DEM file data, see DEM Files (page 634).

**NOTE** SDTS (Spatial Data Transfer Standard (page 2482)) DEMs are not supported in AutoCAD Civil. There is a number of free utilities that you can download and use to convert SDTS data to the supported DEM format.

**To create a surface from DEM data**

1   Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From DEM  .

2   In the Grid Surface From DEM dialog box, browse to the location of the file.

**3** Select the DEM (.dem), GeoTIFF (.tif), ESRI ASCII Grid (.asc or .txt), or ESRI Binary Grid (.adf) file. The filename is displayed in the File Name field.

**4** Click Open.

## Quick Reference

Ribbon

Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From DEM 🏠

Menu

Surfaces menu ➤ Create Surface From DEM

Toolspace Shortcut Menu

Prospector tab: Right-click Surfaces ➤ Create Surface From DEM

Command Line

CreateSurfaceGridFromDEM

Dialog Box

Grid Surface from DEM (standard file selection dialog box)

# Creating a Surface Reference

You can create a reference to an existing surface in a data shortcut (page 2472) or a Vault project.

The surface reference is a lightweight read-only copy of the original, but it gives you access to surface data for other objects such as alignments and profiles. Before you create the surface reference, it must exist in the current project collection on the Toolspace Prospector tab.

**NOTE** A surface reference has no data definition, but has masking and watershed analysis functionality.

**See also:**

■ Creating Data Shortcuts (page 131)

■ Vault: Checking a Drawing in to a Project (page 158)

**To create a reference to a data shortcut surface**

**1** In Toolspace, on the Prospector tab, in Master View, ensure that the correct working folder is identified on the Data Shortcuts node.

**2** Expand the Data Shortcuts ➤ Surfaces collection, right-click the desired surface, and then click Create Reference.

The Create Surface Reference dialog box is displayed, in which you can optionally change properties of the reference surface, as described in the following steps.

**3** In the Create Surface Reference dialog box, change the source of the reference by selecting a source in the Source Surface drop-down list.

**4** Click 🗇 to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

**NOTE** If you do not select a layer, the surface is placed on the current layer.

**5** In the properties grid, click the Value column for the Name property. Enter a name for the surface.

> **NOTE** To name the surface, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**6** To change the style for the surface, select the Style property in the properties grid. Click ⋯ in the Value column.

The Select Surface Style dialog box is displayed. For more information, see Surface Styles and Visualization (page 692).

**7** To change the render material for the surface, select the Render Material property in the properties grid. Click ⋯ in the Value column.

The Select Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578).

**8** Click OK to create the surface reference.

The surface name is displayed in the Surfaces collection in the Prospector tree with a 🖼 next to it.

**To create a reference to a surface in a Vault project**

**1** In Toolspace, on the Prospector tab, in Master View, expand the Projects ➤ <project name> ➤ Surfaces collection, right-click the desired surface, and then click Create Reference.

The Create Surface Reference dialog box is displayed, in which you can optionally change properties of the reference surface, as described in the following steps.

**2** In the Create Surface Reference dialog box, change the source of the reference by selecting a source in the Source Surface drop-down list.

**3** Click 📑 to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the surface is placed on the current layer.

**4** In the properties grid, click the Value column for the Name property. Enter a name for the surface.

> **NOTE** To name the surface, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**5** To change the style for the surface, select the Style property in the properties grid. Click ⋯ in the Value column.

The Select Surface Style dialog box is displayed. For more information, see Surface Styles and Visualization (page 692).

**6** To change the render material for the surface, select the Render Material property in the properties grid. Click ⋯ in the Value column.

The Select Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578).

**7** Click OK to create the surface reference.

The surface name is displayed in the Surfaces collection in the Prospector tree with a 🡕 next to it.

### Quick Reference

Toolspace Shortcut Menu (for data shortcut project)

  Prospector tab: Data Shortcuts ➤ Surfaces ➤ <surface name> ➤ Create Reference

Toolspace Shortcut Menu (for Vault project)

  Prospector tab: Projects ➤ <project name> ➤ Surfaces ➤ <surface name> ➤ Create Reference

Command Line

  CreateSurfaceReference (for Vault surface only)

Dialog Box

  Create Surface Reference (page 2202)

# Adding and Editing Surface Data

You can edit and add data to a surface after you create it.

Surface data categories include:

■  **Boundaries**. Polygons that define outer, hide, and show surface boundary types.

■  **Breaklines**. Includes standard, proximity, non-destructive, and wall breaklines. You can define breaklines from objects in the drawing or import them from a file.

■  **Contours**. Includes contour data that you can define from polyline objects.

■  **DEM files**. Includes USGS Digital Elevation Model DEM files.

> **NOTE** To use SDTS files, first convert them into DEM files. For more information, see About DEM File Data (page 635).

■  **Drawing objects**. Includes Line, Point, Block, Text, 3D Faces, and Polyfaces. Used for creating surface points from AutoCAD entities.

■  **Point files**. Includes ASCII point files.

■  **Point groups**. Includes previously defined point groups.

# Understanding the Surface Definition

A surface definition is a collection of a surface build, data, and edit properties, as well as a list of the operations that you have performed on the surface. The type of definition data items that a surface can have depends on the surface type.

The Definition collection under a surface in the Prospector tree provides access to the surface possible data and edit definition items.

**See also:**

■  Editing and Viewing the Surface Definition (page 681).

### Data Definition

The data definition for a surface comprises the data components that you can add to a surface.

The following table illustrates the data categories that are supported for each surface type. If the surface type does not support a data type, the data type is not available in the surface Definition collection in the Prospector tree.

| Category | TIN Surface | TIN Volume Surface | Grid Surface | Grid Volume Surface |
|---|---|---|---|---|
| Boundaries | Yes | Yes | Yes | Yes |
| Breaklines | Yes | No | No | No |
| Contours | Yes | No | No | No |
| DEM Files | Yes | No | Yes | No |
| Drawing Objects | Yes | No | No | No |
| Point Files | Yes | No | No | No |
| Point groups | Yes | No | No ; | No |

### Edits

Data edits are operations that are not added to any of the existing data components; rather, they are added to a surface definition as edit operations.

For more information about surface edits, including the supported types of edits for the various surface types, see Surface Editing Operations (page 650).

### Operation List

The surface build process is incremental. Every time when you add data to a surface or edit the surface, the surface is updated. When data is removed, the surface is rebuilt. For information about building a surface, see Rebuilding a Surface (page 685).

To support the incremental build process, a surface has an operation list, which is a sequential list of all operations performed on the surface in its current state. For more information about the surface operations, see Surface Editing Operations (page 650) as well as the Surface Properties - Definition (page 2346) tab.

# Boundaries

A boundary is a closed polygon that affects the visibility of the triangles inside it.

When you create boundaries, you specify whether they use arc tessellation (page 615) and non-destructive breaklines (page 615).

### Boundary Types

AutoCAD Civil supports the following types of boundaries:

■ **Outer**. Defines the outer boundary of the surface; all triangles inside it are visible and all triangles that are outside it, are invisible.

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**NOTE** The outer boundary may be affected by subsequent edit operations, such as Add Breaklines. You may need to move the Add Boundary operation to the bottom of the list to achieve the expected result.

---

An outer boundary created using non-destructive breaklines (page 615):
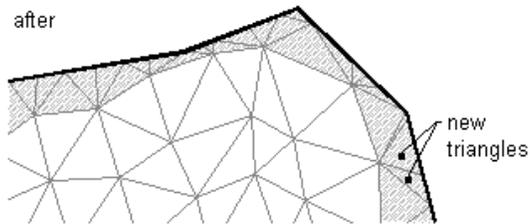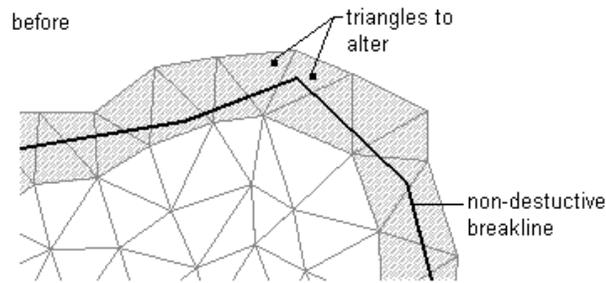
- **Show**. Displays all triangles inside the boundary; you can use it to create visible areas within hide boundaries.

**NOTE** The effect of adding multiple boundaries to a surface is dependent on the order in which they are added. The effects of a boundary can be wholly or partially overridden by a subsequent boundary.

- **Hide**. Masks areas of the triangulation, and therefore contours are not visible in the area; used to punch holes in a surface (for example, a building footprint).
  A hide boundary created using non-destructive breaklines (page 615):

before

building pad
boundary

after

new
triangles
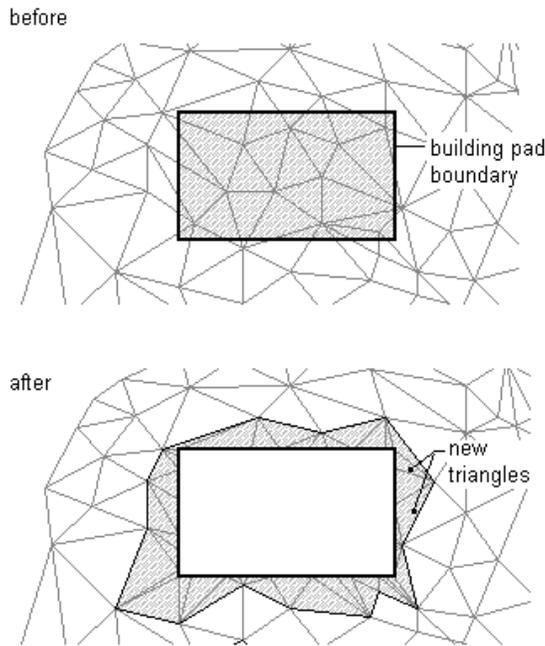
> **NOTE** When you use a hide boundary, the hidden surface area is not deleted. The full surface remains intact. If there are surface TIN lines that you want to remove from the surface, then use the Delete Line operation. For more information, see Deleting TIN or Grid Lines (page 651).

■ **Data Clip**. Creates a surface boundary limited by a polygon object from the drawing, such as 2D and 3D polylines, feature lines, survey figures, parcels, and circles.
You can add more than one data clip boundary to a surface. When you add data, only the most recent data clip boundary is affected. A data clip boundary does not affect the visible part of the surface, but acts as a filter on all data, such as points and breaklines added to the surface after the creation of data clip. For example, if you add a breakline to a surface following the creation of a data clip boundary, only the part of the breakline that is inside the data clip boundary is added.

Surface boundaries are defined by selecting existing polygons from the drawing. The surface definition displays the numerical ID and a list of vertices for each boundary.

## Applying a Data Clip Boundary to Imported Surfaces

You can apply a Data Clip boundary to surfaces created from imported data files, including TIN, LandXML, grading, or corridor surfaces.

**To apply a Data Clip boundary to an imported surface**

1   Create a surface from imported data (TIN, grading, or corridor). See Creating a Surface From a TIN File (page 607) , Creating a New Grading Group (page 741), Creating a Corridor Surface (page 1351).

2   Add the Data Clip boundary to the surface. See Adding Boundaries to a Surface (page 615).

3   In the Prospector tree, expand the Surfaces collection, right-click <surface name> ➤ Surface Properties.

4   In the Surface Properties dialog box, click the Definition tab.

5   Select the Add Data Clip Boundary operation and click ⇪ to move it to the top of the list.

**6** Click Apply.

**7** In the message box, click Rebuild the Surface.

## Adding Boundaries to a Surface

Use the Prospector tree to add boundaries to a surface.

When you add or remove boundaries, the surface is modified and the modification is added to the definition list for the surface.

Boundaries are defined from closed polygons which affects the visibility of the triangles inside them.

**NOTE** If the polygon is not closed, the boundary definition forces a closed polygon.

Areas hidden by boundaries are not included in calculations, such as total area and volume.

When creating a boundary, you set arc tessellation and specify whether the boundary uses non-destructive breaklines when adding boundaries to a surface.

### Arc Tessellation

When you use polygons/polylines that contain curves for boundaries or breaklines, set the mid-ordinate distance of the chord segments, which is used to tessellate the arc segments of the boundary polygon/polyline:

boundary
polygon

chord

added
vertex

mid-ordinate
distance
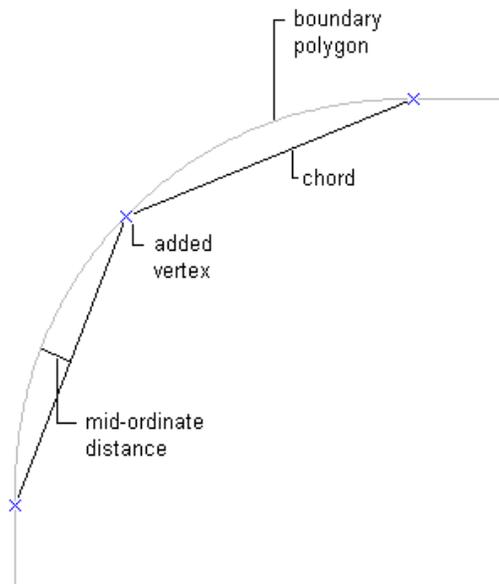
### Non-Destructive Breaklines

Specify whether a boundary uses non-destructive breaklines when you create it by selecting Non-destructive breaklines in the Add Boundaries (page 2359) dialog box.

**NOTE** You cannot add non-destructive breaklines to a grid surface.

When you create a boundary with non-destructive breaklines along the edges of the boundary, the triangle edges are clipped exactly where they cross the boundary:

before
triangles to alter

non-destuctive breakline



after
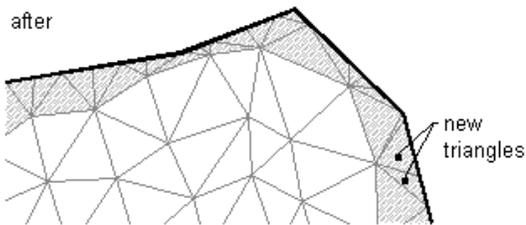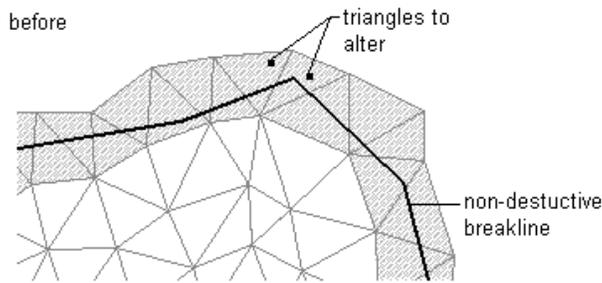
new triangles

If you create a boundary without non-destructive breaklines along the boundary edges, only triangles that are completely within or outside the boundary are affected:



before
triangles to delete

boundary polyline



after
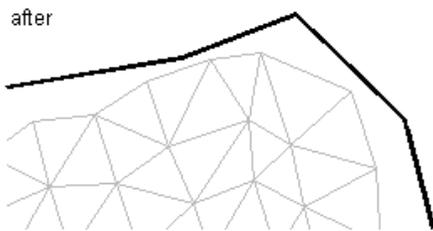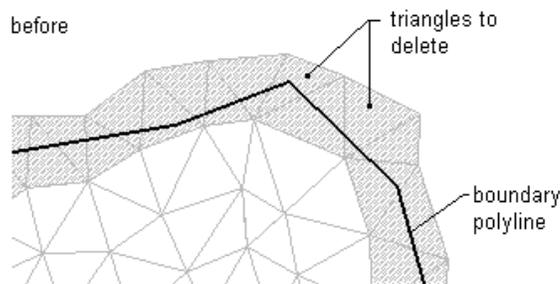
**To add boundaries to a surface**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⬨ , and click Add.

2   In the Add Boundaries (page 2359) dialog box, enter the boundary name in the Name field.

3   Select the boundary type from the Type list. See Boundaries (page 612).

4   Optionally, select Non-destructive breakline specify that the boundary uses non-destructive breaklines.

NOTE  For a TIN surface, by default, the Non-destructive breakline check box is cleared and non-destructive breaklines are not used. For grid surfaces, the Non-destructive breakline field is disabled and not available.

**5** Optionally, if the polygon from which you are creating a boundary has curves, either enter a value in the Mid-Ordinate Distance field or click  to specify a distance in the drawing area.

The Add Boundaries dialog box is closed. You are prompted to select a polyline.

**6** Select one of the following to define the boundary:

- An existing polyline

- An existing polygon, such as a circle object

- An existing parcel

The boundary is created and added to the surface Boundaries collection in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Boundaries ➤ Add

Command Line

AddSurfaceBoundaries

Dialog Box

Add Boundaries (page 2359)

## Inserting Boundary Data into the Drawing

Insert boundaries into the drawing as AutoCAD polylines.

If you deleted the original AutoCAD entity and did not update the surface, you may need to insert boundaries into the drawing. In this case, you can import the boundary from the boundaries collection.

**To insert boundary data into the drawing**

**1** In the Prospector tree, click the surface Boundaries collection.

The boundaries appear in the list view in the Prospector tab.

**2** Right-click a boundary in the list view and click Insert Into Drawing.

- If the boundary does not exist in the drawing, it is added as a polyline object.

- If the boundary exists in the drawing, a message box is displayed stating that no boundaries are inserted into the drawing.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click <boundary-name> in list view ➤ Insert Into Drawing

## Editing Boundary Properties

Use the Prospector tab to view and edit boundary properties.

When you add boundaries to a surface, the icon next to the surface Boundaries collection changes to a ▣
and the boundaries are listed in the Prospector list view.

You can view additional boundary information and perform limited edits to boundary information in the
Boundary Properties (page 2360) dialog box.

**To edit boundary properties**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection and click ⬈ .

   The Prospector list view displays the following:

   ■ The name of the boundary

   ■ The boundary type: Outer, Show, or Hide

   ■ Whether the boundary is trimmed with non-destructive breaklines

2 In the list view, right-click a list item.

3 Click Properties to open the Boundary Properties (page 2360) dialog box where you can edit the name of
   the boundary. All other fields are read only.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Boundaries ➤ right-click <boundary-name>
in the list view ➤ Properties

Dialog Box

Boundary Properties (page 2360)

# Breaklines

Use breaklines to define features, such as retaining walls, curbs, tops of ridges, and streams. Breaklines force
surface triangulation along the breakline preventing triangulation across the breakline.

Breaklines are critical to creating an accurate surface model because it is the interpolation of the data, not
only the data itself, that determines the shape of the model.

**NOTE** You can add breaklines only to TIN surfaces.

The effects of breaklines:

before adding breaklines

after adding breaklines
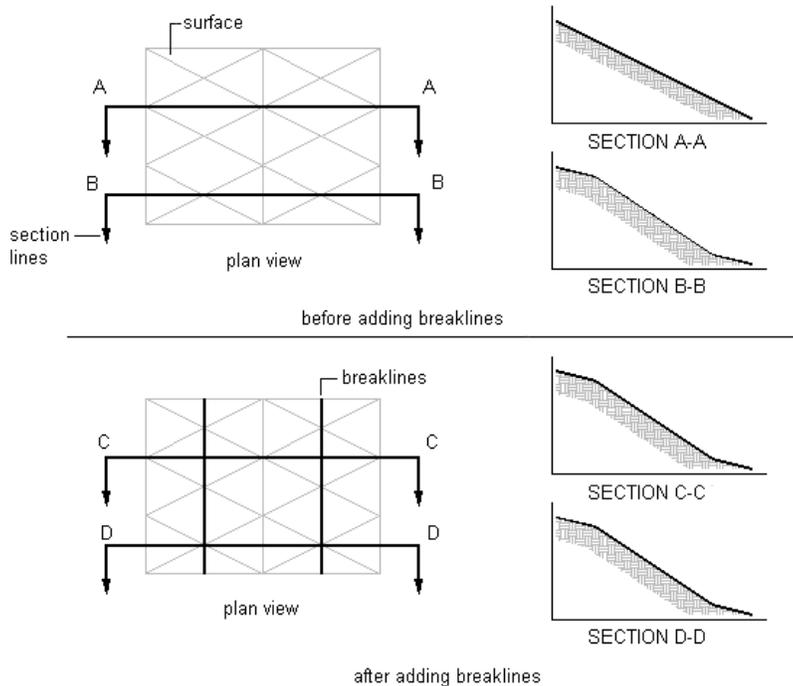
## Types of Breaklines

Define standard, proximity, wall, and non-destructive breaklines for a TIN surface.

■ **Standard**. Defined by selecting 3D lines, grading feature lines, and 3D polylines.

■ **Proximity**. Defined by drawing or selecting a grading feature line or polyline object in the drawing within the extents of the surface boundary. The XYZ coordinates of proximity breakline vertices are determined from the surface TIN points that are in closest proximity to the corresponding vertices of the defining points or entity.

**NOTE** To specify breaklines from points, first convert the points into 3D or 3D polylines using the 'PN command. For more information, see Transparent Commands (page 1581).

■ **Wall**. Defined using grading feature lines, 3D lines, 3D polylines, or by specifying points. A wall breakline is stored as a standard breakline, but defined differently: you provide an offset side for the entire breakline, and an elevation difference for each vertex, or for the entire breakline.

■ **Non-destructive**. Defined using grading feature lines and open or closed AutoCAD objects. A non-destructive breakline maintains the integrity of the original surface.

You can also import breaklines from ASCII FLT files into the surface definition.

**Breakline Definition Options**

When you define a breakline from a polyline with curves, specify a mid-ordinate distance, which is used to tessellate the arcs in the polyline:
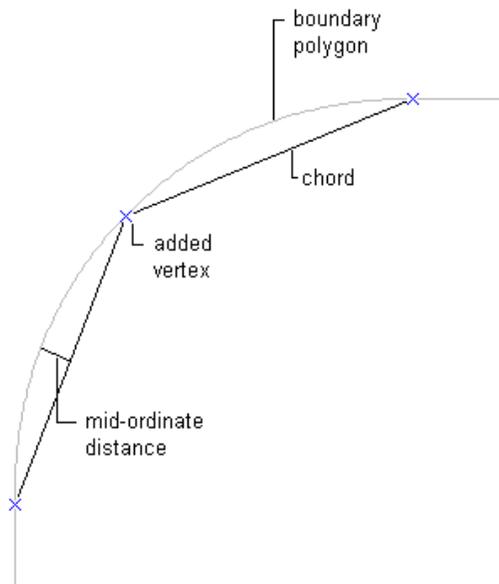
Access global breakline definition options in the Build options property group on the Definition tab (page 2346) of the Surface Properties dialog box.

The breakline definition options include:

■ **Convert Proximity Breaklines**. Automatically converts proximity breaklines to standard breaklines when they are created.

■ **Allow Crossing Breaklines**. Allows breaklines to cross each other.

For more information, see Editing and Viewing the Surface Definition (page 681).

## Creating Standard Breaklines

Use the Add Breaklines dialog box to create standard breaklines.

You can use 3D lines, grading feature lines, and 3D polylines as breaklines. The X, Y, and Z coordinates of each vertex on the polyline that you select are converted into TIN vertices. For 3D lines, each line that you select is defined as a two-point breakline.

**WARNING** If you select a 2D polyline with a zero elevation, then it is saved with that elevation. Use proximity breaklines if you want the elevations calculated automatically.

**To create a standard breakline**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click , and click Add.

2 In the Add Breaklines (page 2360) dialog box, enter the breakline description in the Description field.

3 Select Standard from the Type list.

4 Optionally, if the polyline from which you are creating a breakline has curves, enter a value into the Mid-Ordinate Distance field or  to select a distance in the drawing area.

The Add Breaklines dialog box closes and you are prompted to select a polyline.

**5** Select the polyline or line to define the breakline.

The breakline is created and added to the surface Breaklines collection item in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Breaklines ➤ Add

Command Line

AddSurfaceBreaklines

Dialog Box

Add Breaklines (page 2360)

# Creating Proximity Breaklines

Use the Add Breaklines dialog box to define proximity breaklines that reference surface points in proximity to the vertices of the polyline that you select as the breakline.

Surface points are any surface data points used to create a TIN that are in proximity to the vertices of the polyline that you select as the breakline.

You can define proximity breaklines quickly because you do not have to precisely snap to the surface points that you want to use for the breakline. You can pick locations that are near the points that you want to use. The breakline vertices automatically snap to the nearest surface point when the breakline is added to the surface:
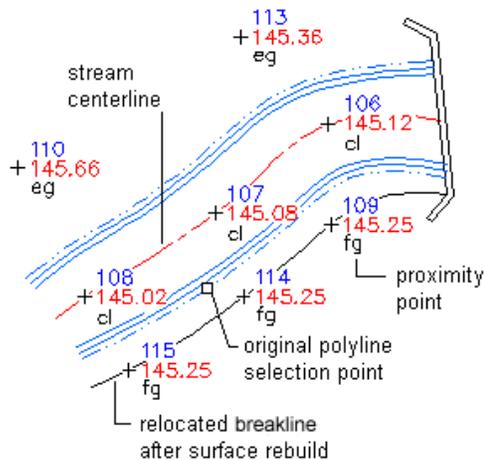


Define proximity breaklines by selecting a polyline. You do not have to draw the polyline precisely between surface points. The breakline definition automatically snaps to the surface point that is nearest each polyline vertex when the breakline is added to the surface.

Proximity breaklines are 2D polylines with elevations of 0. The northing, easting, and elevation are calculated for each vertex according to the closest surface point. By default, proximity breaklines are converted to standard breaklines when added to a surface. You can disable the automatic conversion of proximity breaklines to standard breaklines by setting the Convert Proximity Breaklines To Standard build option to No in the Definition tab (page 2346) of the Surface Properties dialog box.

**To define a proximity breakline**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⬦ , and click Add.

**2** In the Add Breaklines (page 2360) dialog box, enter the breakline description in the Description field.

**3** Select Proximity from the Type list.

**4** Optionally, if the polyline from which you are creating a breakline has curves, enter a value into the Mid-Ordinate Distance field or click ⬦ to digitize a distance in the drawing area.
The Add Breaklines dialog box is closed. You are prompted to select a polyline.

**5** Select one or several polylines or lines to define the breakline.
This creates breaklines and adds them to the surface Breaklines collection in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Breaklines ➤ Add

Command Line

AddSurfaceBreaklines

Dialog Box

Add Breaklines (page 2360)

## Creating Wall Breaklines

Use the Add Breaklines dialog box to define features, such as retaining walls or curbs as breaklines.

By creating wall breaklines, you can more accurately represent the surface. For example, for a retaining wall, you can define the differences in elevation between the material on both sides of the wall in order to represent elevations correctly.

Define wall breaklines by selecting an existing polyline or grading feature line. The polyline is extended by creating new polyline segments and vertices parallel to the original polyline, but offset at an incremented distance to represent the differences in elevation between the material on either side of the wall:
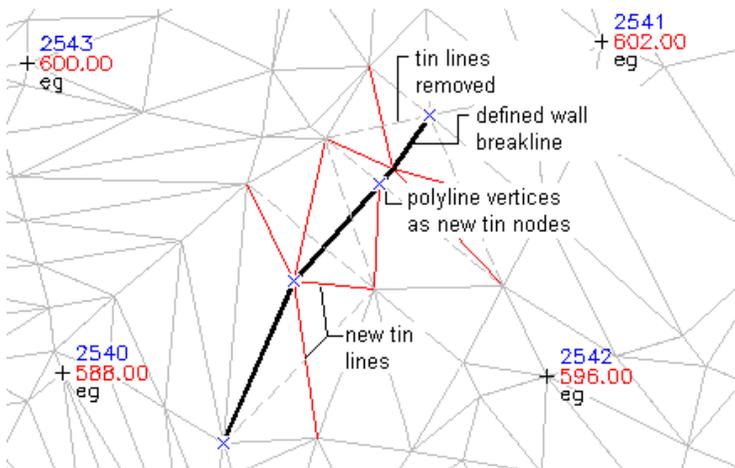
There are two methods used to define wall breaklines:

■ **Define by object vertex (Individual)**. Select a polyline, or line, and select the offset side. For each polyline vertex, you select either the vertex elevation (Existing Vertex Elevation is the default) and the corresponding offset vertex elevation (the default is Existing Polyline Vertex) or the corresponding offset vertex elevation difference. If an elevation difference option is selected, the difference value is the default for each subsequent vertex.

■ **Define by object (All)**. Select a polyline, or line object, and select the offset side. Then, enter the difference elevation to be applied for all offset vertices.

**To define walls or curbs as breaklines**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click  Breaklines, and click Add.

2   In the Add Breaklines (page 2360) dialog box, enter the breakline description in the Description field.

3   From the Type list, select Wall.

4   Optionally, if the polyline from which you are creating a breakline has curves, enter a value into the
    Mid-Ordinate Distance field or click  to digitize a distance in the drawing area.
    The Add Breaklines dialog box is closed. A message prompts you to select a polyline.

5   In the drawing, select the polyline to define the breakline.
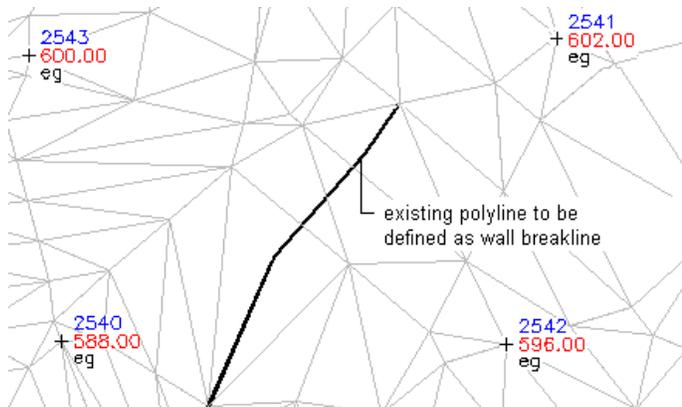


6   Select the offset side for the wall breakline. This is the side for the new offset line that represents the elevation of material on the other side of the wall.
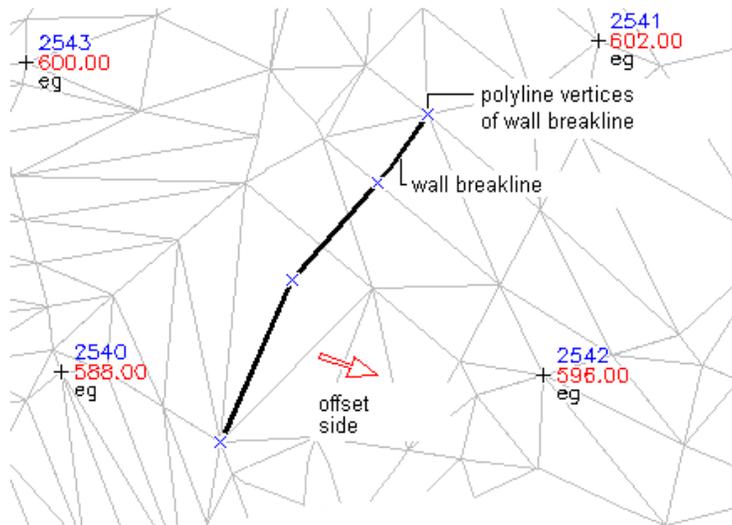
A message prompts you to specify the wall height with one value for all the points (Define By Object) or to specify values for individual points (Define By Vertex).

**7** Do one of the following:

- To define the wall breakline by object, specify **All** and enter the amount to add to elevation at each point.

- To define the wall breakline by object vertex, specify **Individual** at the prompt and enter the elevation or delta for each vertex.
  The prompts provide two ways to define elevations for each offset point: as an elevation (Elevation) or as the elevation difference (Delta) between the control point and the offset point. Deltas may be either positive or negative values.

The breakline is created and added to the surface Breaklines collection in the Prospector tree.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Breaklines ➤ Add

Command Line

AddSurfaceBreaklines

Dialog Box

Add Breaklines (page 2360)

## Creating Non-Destructive Breaklines

Use the Add Breaklines dialog box to create a non-destructive breakline from either a grading feature line or an open or closed AutoCAD object.

When defining a non-destructive breakline, surface points are created at each vertex of the object and at each intersection of a surface triangle edge and the non-destructive breakline object. The new points create additional surface triangles. Non-destructive breaklines are often needed when deleting surface areas where a clean TIN edge does not exist.

The elevation for each new point is extracted from the original surface triangle, therefore maintaining the integrity of the original surface.

**To add non-destructive breaklines**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⬚, and click Add.

2 In the Add Breaklines (page 2360) dialog box, enter the breakline description in the Description field.

3 From the Type list, select Non-destructive.

4 If the object from which you are creating a breakline has curves, enter a value in the Mid-ordinate distance field or click ⬚ to digitize a distance in the drawing area.

5 Click OK.

The Add Breaklines dialog box closes. A message appears prompting you to select an object.

6 In the drawing area, select the objects.

7 Optionally, select other objects or press Enter to end the command.

The breakline is created and added to the surface Breaklines collection in the Prospector tree.

**NOTE** The Description column in the Prospector list view displays the ID of the breakline that was added to the surface. To view information about the breakline vertices, open the Breakline Properties vista. For information, see Viewing Breakline Information (page 627)

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Breaklines ➤ Add

Command Line

AddSurfaceBreaklines

Dialog Box

Add Breaklines (page 2360)

## Importing Breaklines from a File

Use the Add Breaklines dialog box to import breaklines from a file. The supported file format is *.flt* (ASCII file format).

**File Linking Options**

When importing breaklines, you can choose to either maintain a link to the breakline file or break the link:

- **Maintain Link To File**. Reads the breakline from the FLT file when they are added and when the surface is rebuilt. If you edit or delete the source FLT file, the Import Breakline File operation is marked as out-of-date. With this option, you cannot list individual breaklines, view them in the Breakline Properties vista, or import them into the drawing.

- **Break Link To File**. All breaklines in the FLT file are copied into the surface as Add Breakline operations. The FLT file is no longer referenced. When you rebuild the surface, the internal copy of the breakline is

used. The breaklines have the full functionality of other breakline types. You can see the breaklines listed in the Breakline Properties vista, you can pan and zoom to them, and import into the drawing.

**To import breaklines into a drawing**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ![icon], and click Add.

2   In the Add Breaklines (page 2360) dialog box, enter the breakline description in the Description field.

3   Select From File from the Type list.

4   Select the file link option. For information, see File Linking Options (page 625).

5   Click OK.
    The Import Breakline File dialog box is displayed.

6   Select the file to import. It must have an *.flt* extension.
    The breakline is created and added to the surface Breaklines collection in the Prospector tree.

    ---
    **NOTE** The breaklines are imported and grouped by type according to their order in the *.flt* file.
    ---

## Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Breaklines ➤ Add

Command Line

  AddSurfaceBreaklines

Dialog Box

  Add Breaklines (page 2360)

## Creating Breakline Data in a Text File

Use a text editor to create a breakline file manually.

You can then import this text file to use as breakline data for building the surface. For more information, see Importing Breaklines from a File (page 625).

**To create a breakline file manually**

1   Open an ASCII text editor, such as Notepad or Wordpad.

2   Create the file using the following structure:

    ■   Use the (#) symbol as the first character in a comment line.

    ■   Use one of the letters **P**, **S**, **W**, **L**, **R**, **N** (or **ND**) at the beginning of the line to describe the breakline type. These letters stand for Proximity, Standard, Wall, Wall Left, Wall Right, and Non-destructive. Identify a breakline type for each new breakline.

    ■   Enter the X, Y, and Z coordinates separated by a space.

    ■   A description of the breakline can follow the first coordinate of the line.

■ To create another breakline, enter the breakline type letter at the beginning of a line. In the following example, an S is placed at the beginning of line 8 of the file. All the points from this point until the next breakline letter modifier are in one breakline. This breakline is called EOP.

3 Save the file with an *.flt* file extension.

The following is an example of a breakline file:

```
# Autodesk User-Defined Breakline File
P1542.258750 179.318779  0.000000 Flow_Line
 1190.721102 350.527660  0.000000
 860.816542 446.044194  0.000000
 588.600214 619.055276  0.000000
 608.430540 743.406990  0.000000
 565.164367 925.429066  0.000000
S1529.639450 179.318779  100.250000 EOP
 1188.918348 357.736453  101.420000
 1021.998725 330.653391  103.530000
 594.008485 610.044279  102.870000
 761.574952 744.929530  104.780000
 610.233298 754.220180  103.410000
 570.572642 923.626865  100.980000
P1542.258750 179.318779  0.000000 Flow_Line
 1190.721102 350.527660  0.000000
 860.816542 446.044194  0.000000
 588.600214 619.055276  0.000000
 608.430540 743.406990  0.000000
 565.164367 925.429066  0.000000
```

## Viewing Breakline Information

Use the Prospector list view and Breakline Properties vista to view a breakline description, type, and vertices, as well as to zoom or pan to a breakline and its vertices.

**To view breakline in a drawing**

1 Expand the surface Breaklines collection in the Prospector tree.

The breakline operations are displayed in the Prospector tree and list view.

2 Right-click a breakline operation in the Prospector tree or list view and do one of the following:

■ Click Zoom To or Pan To to see the location of the breakline operation in the drawing.

■ Click Properties to display the Breakline Properties (page 2358) vista, which displays the individual breaklines as well as a list of vertices and the coordinates and elevation for each vertex.

3 To zoom or pan to an individual breakline vertex, in the Breakline Properties vista, right-click the vertex and click Zoom To or Pan To.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Breaklines ➤ right-click <breakline-operation-description> ➤ Properties

Dialog Box

Breakline Properties Vista (page 2358)

## Editing Breaklines

Edit any breaklines that you have defined by selecting it in the drawing. You can insert, move, and delete vertices, as well as redefine the elevation at a selected vertex.

Use any AutoCAD editing commands, including:

- Properties command (editing vertex elevations)
- Change command
- Pedit command (graphical editing of polylines)
- AutoCAD grip editing

The surface stores an ID for each of the objects that are defined as breaklines. Therefore, if you modify any of the breakline objects using any of the mentioned commands, the surface is marked as out-of-date until you rebuild it.

**NOTE** You can rebuild a surface automatically by toggling the check mark next to the Rebuild - Automatic menu entry available on the surface shortcut menu in Prospector. For more information, see Rebuilding a Surface (page 685).

**To change a breakline vertex elevation or location**

1 Select the breakline either graphically or by using an AutoCAD command, such as Properties Edit (Pedit) or Change.

2 Perform the edit to the breakline.

   If the breakline and surface definition become out-of-date, a ▽ is displayed next to its node in the Prospector tree.

3 Rebuild the surface to update the breakline definition by right-clicking the surface in the Prospector tree and clicking Rebuild.

**NOTE** You can rebuild a surface automatically by toggling the check mark next to the Rebuild - Automatic menu entry available on the surface short-cut menu in Prospector. For more information, see Rebuilding a Surface (page 685).

**To change the description of a breakline operation**

1 Click the surface Breaklines collection in the Prospector tree.

   The breakline operations with their descriptions and types are displayed in the Prospector list view.

2 Click the breakline operation description in the list view and enter a new description.

**NOTE** You cannot rename individual breakline descriptions. They are named based on the breakline operation description.

## Inserting Breakline Data into a Drawing

Insert breaklines into a drawing as AutoCAD polylines.

**NOTE** You cannot insert into the drawing breaklines that you have imported from an *.flt* file (ASCII file format), unless you used the Break Link To File option. For information, see Importing Breaklines from a File (page 625).

You may need to insert breaklines into the drawing if you deleted the original entity and did not update the surface. In this case, you can import the breakline from the breakline collection.

If any of the selected breaklines exist in the drawing, a message box is displayed stating that no breaklines are inserted into the drawing. Polyline objects are created on the current layer.

**To insert breakline data into the drawing**

1  Click the surface Breaklines collection in the Prospector tree.

The breaklines with their descriptions and types are displayed in the Prospector list view.

2  Right-click a breakline operation in the list view and click Insert to Drawing.

■  If the breakline objects in the breakline operation do not exist in the drawing, they are added to the drawing as polyline objects.

■  If the breaklines exist in the drawing, a message box is displayed stating that no breaklines are inserted into the drawing.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Breaklines ➤ <breakline operation-description> ➤ right-click ➤ Insert To Drawing
Or
Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Breaklines ➤ right-click <breakline-operation-description> in list view ➤ Insert To Drawing
Or
Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Breaklines ➤ <breakline operation-description> ➤ <breakline operation-description: number> in list view ➤ right-click ➤ Insert To Drawing
Or
Breakline Properties vista: Select breakline item in the tree view ➤ right-click ➤ Insert to drawing

# Contours

Use contour data to create or modify a surface.

Surface contour data are created from polylines whose points are all at the same elevation.

You can add contour data only to TIN surfaces.

## Recommendations for Creating Surfaces from Contours

When you create a surface from contour data, especially if you want to add other data to the surface, you can generate a better surface model whose contours will match the contour data if you follow recommended guidelines.

First, create the surface and add all your contour data. For best results, add the contour data in a single operation. Do not add additional data such as spot elevations or breaklines at this time.

You can specify Minimize Flat Areas By options when adding the contour data, or you can add the contour data without the options and then use Check for Contour Problems to locate contour problems and repair

them. For information on repairing contour problems, see Checking for Contour Problems (page 709) and Minimizing Flat Areas in a Surface (page 656).

---

**NOTE** For some contour data sets, Check for Contour Problems report large numbers of errors if you add the contour data to the surface without specifying Minimize Flat Areas By options.

---

After you have added your contour data and repaired problems, verify the contours visually. Add additional contours where needed, and add spot elevation data or breaklines only where they are required to correct a specific problem. For more information on repair, see Supplementing Missing Contour Information (page 631) and Minimizing Flat Areas in a Surface (page 656). Repair all problems.

After you have identified and repaired contour problems, add additional surface data such as additional spot elevation points. Add surface boundaries last. Add boundaries starting at the outside of the surface and working toward the inside.

## Weeding and Supplementing Factors for Contours

Use weeding and supplementing to add or remove vertices along a contour.

Set the weeding and supplementing parameters when you add the contour data to the surface definition in the Add Contour Data (page 2355) dialog box.

### Weeding Factors

Weeding reduces the number of points generated along the contours. The weeding factors ignore both vertices that are closer together than the distance factor and vertices that deflect less than the angle factor. A larger distance and deflection angle weeds a greater number of points. The distance factor is measured in linear units, and the angle factor is measured in angular units. The weeding factors must be less than the supplementing factors.

A point on the contour is weeded by calculating its location in relation to the vertices before and after it. If the length between these three points is less than the weeding length value, and the deflection angle is less than the weeding angle value, then the middle point is not added to the contour data file.

Examples of weeding factor parameters:
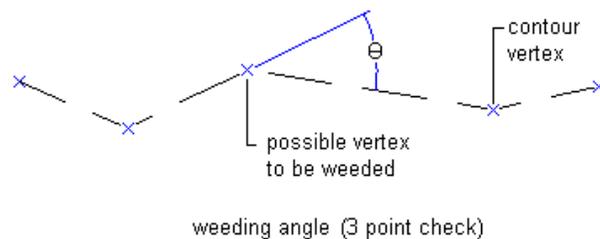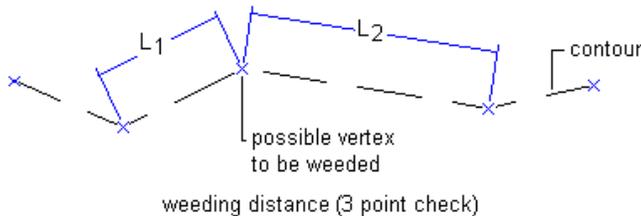
**Supplementing Factors**

Supplementing factors add vertices along contours.

The supplementing distance is the maximum distance between vertices. If the distance between vertices on a contour is greater than the supplementing factor, then points are added along the contour at equal intervals that are less than or equal to the supplementing distance. The smaller the distance, the greater the number of supplemented points.

Example of the distance (SD) parameter with supplemented vertices:
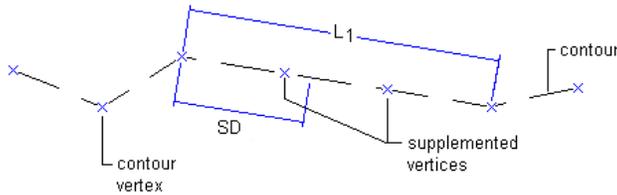


if L$_1$ > supplementing distance, then vertices are added
in equal increments that are less than or equal to SD

The mid-ordinate distance is the distance from the midway point of an arc to the chord of the arc. The mid-ordinate distance is used to add vertices to a polyline curve, creating an approximation of the curve using straight line segments. The length of these segments depends on the value of the mid-ordinate distance.

Example of the mid-ordinate parameter:
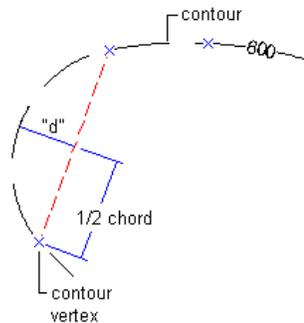


if ratio of "d" to 1/2 chord > supplementing bulge factor,
then vertices will be added on the arc

## Contour Data and Surface Triangulation

Information obtained from a contour map differs greatly from data taken randomly in the field.

Since contour map data is interpolated, the information may be less accurate than direct field data. The accuracy of the final surface model depends on both the quality of the contour map and the contour interval.

If you bring contour data into the drawing as points, then the points would not be very random. In steep areas, points would be close together. In flat areas, there is a greater distance between points on different contours. In both cases, the points along the contours are generally close together. Therefore, information taken from contours does not make effective random point data, and random data points generate the best triangulation.

## Supplementing Missing Contour Information

When you create a model from contours and then generate contours from this model, in some cases, AutoCAD Civil may not generate the same set of contours.

In the most common situation, surface data is missing in either a low or high area, such as near the top of a hill, at the bottom of a valley, or on the edge of the site.

To correct this problem, place spot elevations or another contour line near the place where the surface data is missing. For example, if a contour line is missing along the top of a hill, place a new contour, breakline, or spot elevation at the estimated maximum elevation of the hill. The recommended practice is to place any interpolated data, points, or contours on a separate layer.

If you create a surface using only contour information, you may need to place breaklines or spot elevations in areas where the contour line changes direction drastically. Areas where this is likely to happen include: a crowned roadway, a swale, or a ridge. In these places, contours tend to triangulate onto themselves. In most instances, the Minimize Flat Areas operation resolves this problem. For more information, see Minimizing Flat Areas in a Surface (page 656). In severe situations, you may need to add additional point or breakline data.

For best results, always follow the suggested order of operations for adding contour data to a surface. See Recommendations for Creating Surfaces from Contours (page 629).

## Adding Contour Data to a Surface

Use the Add Contour Data dialog box to add contour data from polylines to a surface.

**To add contour data to a surface**

1  In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click Contours, and click Add.

2  In the Add Contour Data  (page 2355)dialog box, enter the contour description in the Description field.

3  Enter the distance and angle weeding factors in the corresponding fields or click to digitize a distance or angle in the drawing area. For more information, see Weeding and Supplementing Factors for Contours (page 630).

4  Enter the distance and mid-ordinate distance supplementing factors in the corresponding fields or click to select a distance or mid-ordinate distance in the drawing area.

5  Select options for minimizing flat areas in the surface. For more information, see Minimizing Flat Areas in a Surface (page 656).

6  Click OK.

7  Select the polyline to define the contour data and press Enter.

The contour data is created and added to the surface Contours collection in the Prospector tree.

**NOTE** Before displaying the Add Contour Data dialog box, it can be useful to create a selection set of contour objects using the QSELECT command at the command line or choosing Select Similar from the Edit menu.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Contours ➤ Add

Command Line

AddSurfaceContours

Dialog Box

Add Contour Data (page 2355)

## Listing and Identifying Contours

Zoom or pan to contour data and list its description and summary.

**To identify contours in a drawing**

1   Click the surface  Contours collection in the Prospector tree.

The contours with their descriptions and summaries are displayed in the Prospector list view.

2   Right-click a contour data item in the Prospector list view and click Zoom To or Pan To, to see the location of the contour data in the drawing.

## Editing Contours

Edit any contour that you have defined by selecting it in the drawing. You can insert, move, and delete vertices, as well as redefine the elevation at a selected vertex.

You can use any AutoCAD editing commands, including:

■   Properties command (editing vertex elevations)

■   Change command

■   Pedit command (graphical editing of polylines)

■   AutoCAD grip editing

The surface stores an ID for each of the objects that are defined as contours. Therefore, if you modify any of the contour objects using any of the mentioned commands, the surface is marked as out-of-date until you rebuild it.

**NOTE** You can rebuild a surface automatically by toggling the check mark next to the Rebuild - Automatic menu entry available on the surface short-cut menu in Prospector. For more information, see Rebuilding a Surface (page 685).

**To change a contour vertex elevation or location**

1   Select the contour graphically or by using an AutoCAD command, such as PEDIT or CHANGE.

2   Perform the edit to the contour.

If the contour and surface definition become out-of-date, a  is displayed next to its node in the Prospector tree.

3   Rebuild the surface to update the contour definition by right-clicking the surface in the Prospector tree and clicking Rebuild.

**TIP** You can update the surface automatically by turning on Rebuild - Automatic. For more information, see Rebuilding a Surface (page 685).

## Inserting Contour Data Into a Drawing

Insert contour data into the drawing as AutoCAD polylines using the Insert to Drawing command.

In some situations, the surface keeps breakline or contour data (a list of their point coordinates), but does not associate them with an object. This may happen, for example, when you deleted an entity that the surface depends on, but the Copy Deleted Dependent Objects option on the Definition Tab (Surface Properties Dialog Box) (page 2346) page of the Surface Properties dialog box is set to Yes. In this case, the surface simply converts its reference from a drawing entity to a list of points defining the object's geometry.

To restore the reference to the object entity, you can insert the data associated with the deleted contour.

**To insert contour data into the drawing**

1   In the Prospector tree, expand the Surfaces collection ➤ <surface name> ➤ Definition.

2   Click the Contours collection.

    The contours with their descriptions are displayed in the Prospector list view.

3   Right-click a contour operation in the list view and click Insert To Drawing.

    ■   If the contour objects in the contour operation do not exist in the drawing, they are added to the drawing as AutoCAD polyline objects.

    ■   If the contours exist in the drawing, a corresponding message is displayed.

---

**NOTE** You can insert multiple contours using the Insert To Drawing command.

---

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Contours ➤ <breakline operation-description> ➤ right-click ➤ Insert To Drawing
Or
Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ Contours ➤ right-click <breakline-operation-description> in list view ➤ Insert To Drawing

# DEM Files

Use digital elevation model (DEM) files or SDTS files converted to a DEM format to import points into a surface.

DEM files are used to store and transfer large-scale topographic relief information for use in GIS, earth sciences, resource management, land planning, surveying, and engineering projects. DEM files typically contain land XYZ information at a regular grid-spaced interval to represent ground relief.

You can add and remove DEM files, and view DEM information. To transform DEM coordinates so they match the coordinate system of the current drawing when the surface is built, specify a coordinate system for the DEM file.

DEM files are a valuable data source for many planning and engineering tasks that might not need the type of precision gained by doing a ground or aerial survey. DEM files can also, in some places, completely eliminate the need for specific surveys. Usually, the DEM data is not precise enough to use on small-scale studies, but is ideal for large-scale planning and analysis tasks.

Some examples of projects that can benefit from DEM information include hydrologic studies, corridor planning for highways and pipelines, land use planning and analysis, slope analysis, and large-scale project visualization.

### Obtaining DEM Files

DEM files are widely available on the Internet. For example, the United States Geological Survey (USGS) provides many DEM files at their web site.

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**NOTE** SDTS (Spatial Data Transfer Standard) format DEM data is not supported for this release of AutoCAD Civil. However, you can use free utilities to convert SDTS data to DEM format.

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## About DEM File Data

DEM files typically contain land XYZ information at a regular grid-spaced interval to represent ground relief.

DEM files are created to strict standards, as specified by the USGS. For complete information about the DEM file format, see "Standards for Digital Elevation Models," in *National Mapping Program Technical Instructions*, published by the U.S. Department of the Interior, U.S. Geological Survey, National Mapping Division. You can find this documentation on the USGS web site.

In general, the DEM source data is contained in one section of the DEM file (Logical record type `A'), and the actual data (XYZ) information is contained in another part of the DEM file (Logical record type `B').

The following data, based on logical record type `A', is displayed for each DEM file that you add to a surface:

| Element Name | Corresponding Data Element Number in Logical Record Type 'A' | Details |
| --- | --- | --- |
| Description | 1 | The first 140 characters of a DEM file, which may include the file name and location. |
| Estimated Point Total | N/A | The estimated number of points in the DEM file, rounded up to three significant digits. |
| Coordinate System Type | 5 | The ground planimetric reference system: UTM, State Plane, or Lat/Long. |
| Zone | 6 | Code that defines the zone in ground planimetric reference system. These can be found in Appendixes 2-E and 2-F of Standards for Digital Elevation Models. The Zone is meaningful only if Coordinate System Type is UTM or State Plane. |
| Horizontal Datum | 27 | The horizontal datum of the ground planimetric reference system, such as NAD27. |
| Vertical Datum | 26 | ■ 1 = local mean sea level<br>■ 2 = National Geodetic Vertical Datum 1929 (NGVD 29)<br>■ 3 = North American Vertical Datum 1988 (NAVD 88) |

| Element Name | Corresponding Data Element Number in Logical Record Type 'A' | Details |
|---|---|---|
| | | For more information, see Appendix 2-H of the Standards for Digital Elevation Models. |
| Vertical Datum Shift | 31 | The value is the average shift value for the four quadrangle corners obtained from program VERTCON. AutoCAD Civil adds this value to the vertical datum to convert to NAVD88. If the vertical datum is NAVD88, then the vertical datum shift value should be zero. |
| DEM Level | 3 | There are three possible DEM Levels:<br><br>■ Level 1 DEMs are elevation data sets in a standardized format. This level includes 7.5-minute DEMs or an equivalent that is derived from stereo profiling or image correlation of National High Altitude Photography Program, National Aerial Photography Program, or equivalent photographs.<br><br>■ Level 2 DEMs are elevation data sets that were processed or smoothed for consistency and edited to remove identifiable systematic errors. DEM data derived from hypsographic and hydrographic data digitizing, either photogrammetrically or from existing maps, are entered into the Level 2 category after review on a DEM editing system.<br><br>■ Level 3 DEMs are derived from DLG data by using selected elements from both hypsography (contours and spot elevations) and hydrography (lakes, shorelines, and drainage). If necessary, ridge lines and hypsographic effects of major transportation features are also included in the derivation. |
| X-Y Units | 8 | The unit of measure for ground planimetric coordinates throughout the file, such as radians, feet, meters, or arc-seconds. |
| Elevation Units | 9 | The unit of measure for elevation coordinates throughout the file, such as feet or meters. |
| Minimum Elevation | 12 | The minimum elevation for the DEM. The value is the units of measure defined by Elevation Units. |
| Maximum Elevation | 12 | The maximum elevation for the DEM. The value is the units of measure defined by Elevation Units. |
| X Spacing | 15 | DEM spatial resolution for X values. The value is the units of measure defined by the XY Units. |
| Y Spacing | 15 | DEM spatial resolution for Y values. The value is the units of measure defined by the XY Units. |

## Adding DEM Files to a Surface

Use the Prospector tree to add DEM file data to a surface and to work with a DEM file data. The DEM file data is combined with any other surface data that you define.

**To add a DEM file to a surface**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⟨icon⟩, and click Add.

2   In the Add DEM File (page 2356) dialog box, enter the path and name of the DEM file or click ⟨icon⟩ and browse to the location of the DEM file.

3   Select the DEM file so that its name is displayed in the File Name field and click Open.

    The DEM file name and information is displayed in the corresponding Add DEM File dialog box fields.

4   Optionally, change the coordinate system of the DEM data. For more information, see Changing the Specified Coordinate System of a DEM File (page 637).

    The points of the DEM file are transformed from the specified coordinate system of the DEM file to the coordinate system of the current drawing taking into account any Transformation settings specified in the Drawing Settings dialog box.

    **NOTE** DEM files cannot be transformed for grid surfaces. If you include a DEM file as part of a grid surface definition, its coordinate system must match that of the drawing, or it cannot be added.

5   Click OK to import the DEM file into the surface.

The DEM file is added to the surface DEM Files collection in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click DEM Files ➤ Add

Dialog Box

   Add DEM File (page 2356)

## Changing the Specified Coordinate System of a DEM File

Use the Prospector tree to specify or change a DEM file coordinate system. If the DEM file coordinate system is different from the current coordinate system of the drawing, you can specify a coordinate system for the DEM file. The coordinate system you specify for the DEM file should match the data defined in the DEM file itself.

**NOTE** Specify a coordinate system for the DEM file only if the coordinate system of the DEM file is different from the current coordinate system of the drawing and you want a coordinate transformation to occur when you build the surface.

If you used the wrong coordinate system when adding the DEM file, you can select a different coordinate system for the file by changing the DEM file properties.

**NOTE** If the coordinates are transformed, the points no longer lie on a grid. Therefore, the transformation of DEM data is not supported when adding a DEM file to a grid surface. You can change a DEM coordinate system only when adding a DEM file to a TIN surface.

The following table lists some specific examples about how coordinates are read and processed when using various coordinate system settings in your drawings and DEM files:

| When the DEM Coordinate System (source) is set to... | And the Drawing Coordinate System (destination) is set to... | The following results occur: |
| --- | --- | --- |
| NH State Plane NAD27 | NH State Plane NAD83 | The DEM file coordinate data is transformed from NAD27 to NAD83 when you build the surface. |
| NH State Plane NAD27 | None | The DEM file data is not transformed when the surface is built; the DEM file data is taken at face value. For example, point 100,100,100 in the DEM file is point 100,100,100 in the surface. |
| None | None | The DEM file data is not transformed when the surface is built; the DEM file data is taken at face value. For example, point 100,100,100 in the DEM file is point 100,100,100 in the surface. |
| None | NH State Plane NAD83 | The DEM file data is not transformed when the surface is built; the DEM file data is taken at face value. For example, point 100,100,100 in the DEM file is point 100,100,100 in the surface. |
| NH State Plane NAD83 | NH State Plane NAD83 | No transformation is applied.<br><br>**NOTE** Specifying a coordinate system for a DEM file is not necessary in this situation. However, if you are working with multiple drawings in the current project, and the drawings have different coordinate systems, it is recommended that you specify a coordinate system for the DEM file in case you need to rebuild the surface in a drawing that has a different coordinate system. |

**To change the specified coordinate system of a DEM file**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, and click 🏠 DEM Files.

2 Right-click a DEM file in the Prospector list view and click Properties.

3 In the DEM File Properties (page 2356) dialog box, select the CS Code property field and click the browse button.

4 In the Select Coordinate Zone (page 2366)dialog box, click the Category list and select a category.

**5** Select a coordinate system from the category list.

**6** Optionally, click Properties to view the properties of the selected coordinate system.

**7** Click OK to select the coordinate system and close the dialog box.

The DEM file property group displays the newly selected coordinate system.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ DEM Files ➤ right-click <DEM-file> in list view ➤ Properties

Dialog Box

DEM File Properties (page 2356)

## Importing DEM Extents

You can import the extents of a selected DEM file into the drawing as 2D polyline objects.

The polyline is drawn as a rectangle representing the four corners of the DEM extents.

**To import DEM extents**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, and click  .

The DEM files with their names, modified dates, and sizes are displayed in the Prospector list view.

**2** In the list view, right-click the DEM file for which you want to import the extents and click Import DEM Extents.

The DEM extents are drawn as a polyline rectangle representing the four corners of the DEM extents.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ DEM Files ➤ right-click <DEM-file> in list view ➤ Import DEM Extents

## Viewing DEM Data and Properties

Zoom or pan to DEM file data, and view information about its data including its coordinate system, projection, and datum.

**To zoom or pan to DEM file data**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection and click  DEM Files.

The DEM files are displayed in the Prospector list view.

**2** In the list view, right-click a DEM file and click Zoom To or Pan To.

The drawing area displays the surface data zooming or panning to the area relative to the selected DEM data (according to the surface style settings). For information about setting and changing the surface display, see Surface Styles and Visualization (page 692).

**To view DEM file properties**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, and click ⊞ .

The DEM files are displayed in the Prospector list view.

**2** In the list view, right-click on the DEM file and click Properties to display the DEM File (Add/Properties) Dialog Box (page 2356).

Detailed information about the DEM file is displayed in the DEM File Information field. For more information, see About DEM File Data (page 635).

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ DEM Files ➤ right-click <DEM-file> in list view ➤ Properties

## DEM Files and Memory Usage

Large TIN surfaces, such as those built from DEM files, can require a substantial amount of memory (RAM and virtual memory) and disk space to build and save the surface. TIN surfaces require about 100 bytes per point, whereas grid surfaces require about 16 bytes per point.

It is important to ensure that you have sufficient resources before you build a surface using a DEM file. Also, be aware that certain commands may take longer than expected when working with these surfaces.

### General Guidelines for DEM File Memory Usage

As a general rule, a 1 MB DEM file contains 160,000 points. For larger DEM files, the number of points grows proportionately to the DEM file size at a rate of approximately 160,000 points per 1 MB of DEM file size.

**See also:**

■ Working with Large Surfaces (page 689)

# Adding Surface Point Data from AutoCAD Drawing Objects

Use AutoCAD objects such as 3D lines to add surface point data to the surface definition.

The selected AutoCAD objects are interpreted based on their points, and specific point data is created for each object type that you add to the surface definition.

For several object types, you can specify whether to maintain the object edges when adding the object points to AutoCAD Civil.

**NOTE** Drawing objects added to the surface are added as point data. After the point data has been added to the surface there is no relationship between the surface and the original drawing objects.

You can select the following AutoCAD drawing objects to add points to a surface definition:

■ **Points**. The XYZ coordinates of the object are used to define the surface point.

■ **Lines**. The XYZ coordinates of the object endpoints are used to define surface points.

■ **Blocks**. The block insertion point XYZ coordinates are used to define the surface point.

■ **Text**. The text insertion point XYZ coordinates are used to define the surface point.

- **3D Faces**. The XYZ coordinates of the object endpoints are used to define surface points.

- **Polyfaces**. The XYZ coordinates of the object endpoints are used to define surface points.

> **NOTE** Optionally, when adding lines, 3D faces, and polyfaces, you can maintain the original AutoCAD object edges as surface triangle edges.

> **NOTE** To add COGO points to surface data, add point groups. For more information, see Point Groups (page 644).

**To add points to a surface from AutoCAD objects**

1  In the Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⌂, and click Add.

2  In the Add Points From Drawing Objects (page 2362) dialog box, select the object type from the Object Type list.

3  Click Maintain Edges From Objects if you are adding points from lines, 3D faces, or polyfaces, and you want to maintain the AutoCAD object edges. This prevents AutoCAD Civil from optimizing the edges.

4  In the Description field, enter a description for the object.

5  Click OK.

6  Select the object in the drawing or use any of the standard AutoCAD object selection methods.

The objects are added to the surface definition and are displayed in the surface Drawing Objects collection list view with their description and type.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Drawing Objects ➤ Add

Dialog Box

Add Points from Drawing Objects (page 2362)

# Moving Blocks to Attribute Elevation

Move AutoCAD block objects with attributes (such as labels or tags) at their insertion point to the elevation of one of the block's attribute objects.

Use this command when you need to use block objects in the drawing as surface point data.

For example, the block object can display an elevation value from one of its attributes, but the elevation of the block reference can be different. Using this command prepares the surface data before defining a surface from block reference objects using the Add Points From Drawing Objects (page 2362) command.

**To move block objects to their attribute elevation**

1  Select a surface. On the Surface tab, click Surface Tools panel ➤ ▼ ➤ Move Blocks To Attribute Elevation 🖼 .

2  In the Move Blocks to Attribute Elevation dialog box, select a block reference name from the list or click  to select a block reference from the drawing.

3  Under Select Elevation Attribute Tag, select the required tag.

4  Click OK.

The selected block reference objects are moved to the elevation of the specified attribute.

### Quick Reference

Ribbon

Select a surface. On the Surface tab, click Surface Tools panel ➤ ▼ ➤ Move Blocks To Attribute Elevation 

Menu

Surfaces menu ➤ Utilities ➤ Move Blocks To Attribute Elevation

Command Line

MoveBlocksToAttribElev

Dialog Box

Move Blocks to Attribute Elevation (page 2370)

## Moving Text to Elevation

Move AutoCAD text objects at their insertion point to the elevation of their numeric text values.

Use this command when you need to use a TEXT object in the drawing as surface point data.

You can select AutoCAD numeric TEXT and MTEXT objects in the drawing and move each object at its insertion point to the elevation corresponding to its numeric text value.

**To move a text object to the elevation value**

1  Select a surface. On the Surface tab, click Surface Tools panel ➤ ▼ ➤ Move Text To Elevation .

2  Select the text object in the drawing or use any of the standard AutoCAD object selection methods.

3  Click Enter.

If the value of a text object is a real number, the text object is moved to the elevation corresponding to the numeric text value.

### Quick Reference

Ribbon

Select a surface. On the Surface tab, click Surface Tools panel ➤ ▼ ➤ Move Text To Elevation 

Menu

Surfaces menu ➤ Utilities ➤ Move Text To Elevation

Command Line

MoveTextToElevation

# Point Files

If you have point data files that you created either manually or by downloading a data collector, you can use the point data as surface data.

Any point file format that can be imported to create AutoCAD Civil points can be used to add points to surfaces.

The following is an example of the PENZ space-delimited format, which is just one of the point file formats recognized by the surface Add Point File command.

A PENZ space-delimited point file contains: point number, easting (X), northing (Y), elevation (Z)). Within the file, the following syntax is used for each point:

```
[Point Number] [Easting (or X)] [Northing (or Y)] [Elevation (or Z)]
```

An example of the contents of a PENZ space delimited format point text file:

```
26303 315872.971714  4838799.938284      111.250000
26304 315876.474880  4838813.524640      111.280000
26305 315873.911320  4838829.056520      111.950000
26306 315861.792509  4838799.287159      112.060000
26307 315855.883079  4838798.942969      114.160000
26308 315855.585197  4838811.389822      114.080000
26309 315861.566272  4838812.001057      111.920000
```

There are several supported point file formats. If your point files use a format that is not recognized by the surface Add Point File command, you can convert the files to a supported format.

First, you create a point file format that describes the layout of the data in the point files. Then, using the point file format, you convert the files to a supported format using the Transfer Points utility. For more information, see Transferring and Converting Point Data (page 506).

You can manually create ASCII text files that contain point data. For more information, see Creating a Point File Manually (page 644).

## Adding Surface Data from a Point File

Add points from a point file directly to an existing surface.

**To add point files to a surface**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ⊞ , and click Add.

2   In the Add Point File (page 2361) dialog box, select the format of the point file or click ✐ to create a point file format that describes the layout of the data in the point data file.

3   Enter the point file path and name into the Source File field or click ✚ to browse and select the point file.

4   Click Open to add the point file to the surface.

    If the point file contains a line with bad data, an error (including the line number) is posted the Event Viewer and the operation continues.

    The point file is added to the Point Files item view.

**Quick Reference**

Toolspace Shortcut Menu

 Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Point Files ➤ Add

Command Line

 AddPointFile

Dialog Box

 Import Point File (page 2124)

## Creating a Point File Manually

Create a PENZ space-delimited point file containing point information and use the file to create or add data to a surface in AutoCAD Civil.

**To create a point file manually**

1  Open a text editor, such as Notepad.

2  Enter the point number, easting, northing, and elevation values. Separate each value by one or more spaces.

3  Enter the information for each point on a separate line. For best results, do not include extra spaces at the end of a line or blank lines at the end of the file. The file should consist entirely of point data. However, you can add a comment line by entering a # sign at the beginning of a line.

4  Save the file as a text file.

An example of a PENZ space-delimited point file:

```
# Autodesk Point File Format
3 379.910000    511.270000    227.620000
4 393.880000    497.100000    225.750000
5 382.640000    464.050000    223.890000
6 403.690000    429.340000    222.880000
103 190.080000   307.560000   194.250000
104 182.090000   322.940000   194.940000
105 174.100000   338.320000   195.630000
106 142.910000   340.320000   203.030000
107 117.470000   342.780000   208.460000
108 124.430000   394.400000   208.410000
109 137.700000   400.690000   205.430000
```

# Point Groups

Add point groups to a surface definition.

Use point groups to break the drawing COGO points into smaller groupings of points, to identify points that share common characteristics, such as number, name, elevation, or that are used to perform a task, such as creating a surface.

You can create point groups that contain specific points, such as all existing ground points, making it easier to manage the surface points.

See Understanding Point Groups (page 521).

## Adding Point Groups

Use the Point Groups collection of the Prospector tree to add point groups to a surface definition.

For information about creating point groups, see Creating Point Groups (page 529).

**To add a point group to a surface**

1  Create a point group if needed. For information, see Creating Point Groups (page 529).

2  In Toolspace, on the Prospector tab, right-click the surface [⊕] Point Groups collection and click Add.

3  In the Point Groups dialog box, in the list of available point groups, select the point group to add to the surface.

4  Click OK to add the point group to the surface.

The point group is added to the Point Group list view in Prospector.

---

**NOTE** For information about other point group functionality, such as viewing and changing the properties of a point group, see Understanding Point Groups (page 521).

---

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Point Groups ➤ Add

Command Line

AddSurfacePointGroup

Dialog Box

Point Groups (page 2118)

# Moving Blocks to a Surface Elevation

Use the Move Blocks to Surface command to manipulate reference object block data.

You can use the reference object block data to define a surface. Moving blocks to surface can also be useful for visualization of multi-view block objects.

**To move Block Reference objects to a surface elevation**

1  Select a surface. On the Surface tab, click Surface Tools panel ➤ ▾ ➤ Move Blocks To Surface 🖱 .

2  If the drawing contains more then one surface, from the Select a Surface dialog box, select the surface or click 🖱 to pick the surface manually.

3  In the Move Blocks to Surface dialog box, select the Block Reference objects or click 🖱 to select the block reference objects directly from the drawing.

4  Click OK.

Each selected Block Reference object is moved from its current elevation to the elevation of the surface at the object insertion point.

**Quick Reference**

Ribbon

Select a surface. On the Surface tab, click Surface Tools panel ➤ ▼ ➤ Move Blocks To Surface 🖼

Menu

Surfaces menu ➤ Utilities ➤ Move Blocks To Surface

Command Line

MoveBlocksToSurface

Dialog Box

Move Blocks to Surface (page 2369)

# Extracting Surface Data

You can extract AutoCAD primitive entities from an AutoCAD Civil surface.

Extracting individual components from a surface enables you to manipulate these components without exploding the surface object. The surface data available to extract is based on the display properties of the selected surface.

**NOTE** The extracted objects preserve the properties set in the current surface style.

The following table lists the extracted objects for each type of surface components.

| Surface component ... | Is extracted as ... |
|---|---|
| Points (2D and 3D) | Point objects |
| Triangles (2D and 3D) | 3D face objects |
| Border (2D) | ■ POLYLINE object<br>■ LWPOLYLINE object (if the Border Display Mode (page 2335) is set to Flatten to Elevation) |
| Border (3D) | ■ POLYLINE object<br>■ LWPOLYLINE object (if the Border Display Mode (page 2335) is set to Flatten to Elevation)<br>■ 3D face object (if Use Datum (page 2335) and Project Grid to Datum (page 2335) are set to true) |
| Major Contour (2D and 3D) | LWPOLYLINE object |
| Minor Contour (2D and 3D) | LWPOLYLINE object |
| User Contours (2D and 3D) | LWPOLYLINE object |
| Gridded (2D and 3D) | ■ POLYLINE object (if the Border Display Mode (page 2335) is set to Use Surface Elevation<br>■ LWPOLYLINE object (if the Border Display Mode (page 2335) is set to Flatten to Elevation) |

| Surface component ... | Is extracted as ... |
|---|---|
| Directions (2D and 3D) | The type of object depends on the surface style settings. See Creating a Surface Style (page 693). |
| Elevations (2D and 3D) | |
| Slopes (2D and 3D) | |
| Slope Arrows (2D and 3D) | |
| Watersheds (2D and 3D) | |

**To extract an object from a surface**

1  Select a surface. On the Surface tab, Surface Tools panel, click Extract Objects  .

2  If more than one surface is available in the drawing, in the Select a Surface dialog box, select the surface from which to extract components.

3  In the Extract Objects from Surface dialog box, under Property, deselect the components that you do not want to extract (all visible surface components are selected by default).

4  Under Value, do one of the following:

   ■ Choose Select All to select all the surface components for extraction.

   ■ Choose Select From Drawing, click  , and select the required surface components.

5  Click OK.

   Every selected surface component is extracted from the surface object.

## Quick Reference

Ribbon

Select a surface. On the Surface tab, Surface Tools panel, click Extract Objects 

Menu

Surfaces menu ➤ Utilities ➤ Extract Objects From Surface

Command Line

SurfaceExtractObjects

Dialog Box

Extract Objects from Surface (page 2369)

# Surface Errors and Issues

Identify and troubleshoot errors or problems with a surface, or its data components.

# Definition Type Errors

View several error types that may be displayed in the surface's Operation Type list on the Definition tab of the Surface Properties dialog box.

The error types are:

- **Item modified** ⚠ Indicates that the definition item has been modified after being added to the surface.

- **Item not found** ⚠ Indicates that the definition item is not found when the surface properties dialog box is initialized.

- **Item not OK** 🏠⊘ Indicates that the definition item was not successfully added due to a change in the surface.

For more information about the Operation Type list, see (page 2346)

# Build Errors

Use the Event Viewer to view surface build errors that occur when data or edit operations cannot be performed due to removal of the data or its dependencies.

The following is a list of build errors that can be displayed:

| Error String | Possible Cause |
|---|---|
| Build Error - Duplicate Point | A duplicate point XY location was ignored at <XY co-ordinates>. |
| Build Error - Crossing Break-line | A crossing breakline was ignored at <starting or ending coordinate>. |
| Build Error - Breakline Cross-ing an Existing Data Point | Breakline crosses an existing point. |

# Surface Disconnection Error

Use the Surface Properties dialog box to view and correct a surface disconnection error, which can occur when a surface becomes disconnected from an external file that it references.

When this occurs, ⚠ is displayed in the Import Surface operation type on the (page 2346) of the Surface Properties dialog box indicating that the item was not found. You cannot rebuild the surface. However, additional data can be added, and certain edits (those that do not cause a rebuild) can be performed.

The problem can be fixed by editing the Operation Type parameter and entering the correct path and name for the external file.

# Unable to Rebuild Snapshot

An exclamation icon ⚠ is displayed next to the definition of the surface in the Prospector tree view and you are unable to rebuild the surface snapshot.

When you rebuild the surface snapshot, the following error message is displayed:

*Unable to rebuild Surface. Couldn't find file(s).*

If you installed AutoCAD Civil to a location other than the default, you must update the XML file's path before you can update the snapshot.

**To locate and update the XML file path**

1  To update the path, right-click the surface name in the Prospector tree, right-click, and click Properties.

2  In the Surface Properties dialog box, on the Definition tab, click the folder icon next to Import XML File.

3  In the Import XML File dialog box, browse to the location where you installed AutoCAD Civil, and locate the XML files.

4  Click Open. Click OK.

5  When prompted if you want to rebuild the surface, click Yes to update the surface properties, and rebuild the surface.

6  To update the snapshot, right-click the surface name in the Prospector tree, and click Rebuild Snapshot.

7  Right-click the surface name in the Prospector tree, and click Rebuild.

# Using the Default Surface Settings

When you import a surface TIN or XML file, the surface uses the default styles as specified in the surface settings.

To use a different style, after you import the file, change the assigned style by editing the surface properties:

1  In the Prospector tab, right-click the surface and click Properties.

2  Click the Information tab and select the desired Object Style from the drop-down list.

3  Click OK to apply the change and close the dialog box.

# Deleting AutoCAD Objects from a Surface

When trying to remove AutoCAD drawing objects using the Erase command, the surface does not reflect the change.

**To delete drawing objects from a surface definition**

1  In the Prospector tab, right-click the surface to be modified. Click Properties.

2  In the Definition tab, under Operation Type, clear the Drawing Objects check box, or you can right-click Drawing Objects and click Remove From Definition.

3  When prompted to rebuild the surface, click Yes.

The drawing objects are removed from the surface definition, and the surface is automatically updated.

# Surface Analysis Values are Not Updated

After modifying surface data, the values in the ranges of the Analysis tab of the Surface Properties dialog box are not updated. Surface legend tables are not updated either.

Using the Analysis tab of the Surface Properties dialog box, you can create ranges for slopes, elevations, directions, watersheds, and contours. These ranges require manual recalculation before the surface display or the legend table can be updated. If the surface data changes, open the Surface Properties dialog box and click the Analysis button in the Ranges area of the Analysis tab to reprocess the data. Surface legend tables that are set up to be dynamic are then automatically updated with the changes.

# Surface Editing Operations

Edit surfaces using data edit operations, which are added to a surface definition as edit operations and not to any of the existing surface data components.

You can access the data editing functionality for a surface by right-clicking ![edit icon] Edits under a surface Definition in the Prospector tree. If a surface type does not support an edit operation, the operation is not listed. The following table lists the edits that are supported for each surface type:

| Edit Operation | TIN Surface | TIN Volume Surface | Grid Surface | Grid Volume Surface |
|---|---|---|---|---|
| Add points | Yes | Editing operations are not supported | Yes | Editing operations are not supported |
| Delete points | Yes | | Yes | |
| Modify points | Yes | | Yes | |
| Move point | Yes | | No | |
| Add line | Yes | | No | |
| Delete line | Yes | | Yes | |
| Swap edge | Yes | | No | |
| Minimize flat areas | Yes | | No | |
| Smooth surface | Yes | | No | |
| Paste surface | Yes | | No | |
| Raise/Lower surface | Yes | | Yes | |
| Simplify surface | Yes | | No | |

# Adding TIN Lines

Use the Prospector tree to add new TIN lines to an existing surface. This operation modifies the way that the surface is triangulated.

The endpoints of a new line must fall at the endpoints of other surface lines. If the new line passes through one or more existing surface lines, the surface retriangulates that area.

---

**TIP** To perform this operation, ensure that the surface style has triangles visibility enabled. See Surface Styles and Visualization (page 692).

---

**NOTE** TIN volume surfaces do not support editing operations.

---

**To add TIN lines**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ![edit icon] , and click Add Line.

**2** At the command line, you are prompted to select the endpoints for the new TIN line.

**3** Click existing vertices as the endpoints for the new TIN lines.

The triangulation of the surface is modified to reflect the new lines.

**4** Optionally, continue to select endpoints to add additional new lines.

**5** Press Enter to complete the operation.

The new TIN lines are added as Add Line operations to the Edits list view in Prospector.

---

**NOTE** The Description column in the list view displays the coordinates of the selected points that created the TIN line.

---

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Add Line

Command Line

AddSurfaceLine

# Deleting TIN or Grid Lines

Use the Prospector tree to delete TIN or grid lines from a surface.

---

**NOTE** TIN or grid volume surfaces do not support editing operations.

---

Use this command to remove:

■ Long and narrow TIN triangles on the perimeter of a surface.

■ TIN or grid lines within a pond or building foundation to create a void area to prevent contours from being drawn through the void areas.

When you remove an edge, an interior border, following the adjacent lines, is created, or the exterior border is modified to follow the new lines.

---

**TIP** To perform this operation, ensure that the surface style has triangles visibility enabled and that the OSNAP Endpoint setting is disabled. See Surface Styles and Visualization (page 692).

---

**To delete TIN or Grid lines**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click , and click Delete Line.

**2** At the command line, you are prompted to select the edges (lines) to remove.

**3** Click the edges that you want to remove and press Enter.

---

**NOTE** If you are deleting grid lines, select the grid lines, not the triangulation lines.

---

The edges are removed. An interior border, following the adjacent TIN or grid lines, is created, or the exterior border is modified to follow the new TIN or grid lines.

The edits are added as Delete Line or Delete Multiple Lines operations to the Edits list view in Prospector.

**NOTE** The Description column in the list view displays the coordinates of the vertices for the edge that was deleted.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Delete Line

Command Line

DeleteSurfaceLine

# Swapping Edges

Use edge swapping to change the direction of two triangle faces in the surface model so that, for example, the triangle edges match ridges or swales.

**NOTE** TIN volume surfaces do not support editing operations.

To swap an edge, observe the following conditions:

■ You must click within 1 unit of an edge.

■ Two visible triangles are separated by the edge.

■ The edge is not a breakline edge or an edge created by an Add Line operation.

■ The quadrilateral formed by the two triangles (which are separated by the edge) is convex.

**TIP** To perform this operation, ensure that the surface style has triangles visibility enabled. See Surface Styles and Visualization (page 692).

**To swap an edge**

1  In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click 🖉 , and click Swap Edge.

2  At the command line, you are prompted to select an edge (line) to swap.

3  Click the edge to swap it.
   The edge is swapped if the criteria are met. For information, see Swapping Edges (page 652).

4  Optionally, continue to select edges to swap by clicking them.

5  Press Enter to complete the operation.
   The edits are added as Swap Edge operations to the Edits list view in Prospector.

   **NOTE** The Description column in the Prospector list view displays the coordinates of the point that you picked in the drawing area to select the edge.

### Quick Reference

# Adding Points to Surfaces

Use the Prospector tree to add points directly to a surface model.

**NOTE** TIN or grid volume surfaces do not support editing operations.

Adding points in this manner constitutes a surface edit, and can be enabled and disabled in the surface properties. This operation adds points to the surface definition rather than to the surface data components.

**NOTE** If you are adding a point to a grid surface, you can add points only outside the existing border or to a hole with no points inside the surface area. This command uses the SNAP orientation (based on the spacing and orientation of the surface being edited).

**TIP** To perform this operation, ensure that the surface style has triangles visibility enabled. See Surface Styles and Visualization (page 692).

**To add points**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click  , and click Add Point.

2 At the command line, you are prompted to select a point.

3 Click on the location for the point.
  You are prompted for the elevation of the point.

4 Specify the elevation at the command line.

5 Optionally, continue to select other points to add, by clicking their locations.

6 Press Enter to complete the operation.
  The points are added as Add Point or Add Grid Point operations to the Edits list view in Prospector.

  **NOTE** The Description column in the list view displays the coordinates and elevation for the points that were added.

### Quick Reference

# Deleting Surface Points

Use the Prospector tree to delete points from a surface and remove inaccurate or unnecessary data.

**NOTE** TIN or grid volume surfaces do not support editing operations.

The points are not deleted from any of the existing surface data components. The deleted point operations are added only to the surface definition.

**TIP** To perform this operation, ensure that the surface style has triangles or points visibility enabled. See Surface Styles and Visualization (page 692).

**To delete points**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click, and click Delete Point.

2 At the command line, you are prompted to select points.

3 Click the points you want to delete from the surface.

4 Press Enter.

5 The selected points are deleted from the surface.

**NOTE** When you delete an interior point from a grid surface, an interior border, following the adjacent grid lines, is created.

6 Optionally, continue to select other points to delete by clicking them.

7 Press Enter to complete the operation.

The deleted points are added as Delete Point, Delete Multiple Points, or Delete Grid Point operations to the Edits list view in Prospector.

**NOTE** The Description column in the list view displays the coordinates and elevation for the points that were deleted.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Delete Point

Command Line

DeleteSurfacePoint
DeleteSurfaceGridPoint

# Modifying Surface Points

Use the Prospector tree to change the elevations of individual surface points.

**NOTE** TIN or grid volume surfaces do not support editing operations.

These points are not modified in any of the existing surface data components. The modify point operations are added only to the surface definition.

To perform this operation, ensure that the surface style has points visibility enabled.

See Surface Styles and Visualization (page 692).

**To modify the elevation of single surface points**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ✎, and click Modify Point.

**2** At the command line, you are prompted to select the points to edit.

**3** In the drawing area, select the points.

**4** Press Enter.

**5** You are prompted to specify the new elevation for the points.

**6** Specify the elevation at the command line.

**7** Optionally, continue to select other points to modify.

**8** Press Enter to complete the operation.

**9** The modified points are added as Modify Point or Modify Multiple Points operations in the Edits list view in Prospector.

---

**NOTE** The Description column in the list view displays the coordinates and new elevation (Z) for the points that were modified.

---

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Modify Point

Command Line

EditSurfacePoint

## Moving Surface Points

Use the Prospector tree to move an existing surface point to a new location. The surface triangulation is updated accordingly.

---

**NOTE** TIN or grid volume surfaces do not support editing operations.

---

The points you move are not modified in any of the existing surface data components. The move point operation is added to the surface definition. This operation does not require the surface points to be visible.

**To move surface points**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ✎, and click Move Point.

**2** At the command line, you are prompted to select the point you want to move.

**3** In the drawing area, click the point.

You are prompted to select the new location for the point.

**4** Click the new location in the drawing.

**5** Optionally, repeat steps 2 and 3 or press Enter to end the command.

The moved points are added as Move Point operations to the Edits list view in Prospector.

---

**NOTE** The Description column in the list view displays the coordinates of the old (from) and new (to) location of the point.

---

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Move Point

Command Line

MoveSurfacePoint

## Minimizing Flat Areas in a Surface

When contour data is added to a TIN surface, the resulting surface can contain flat spots (triangles whose points all come from the same contour) and flat edges (triangle edges that join points from the same contour or from different contours at the same elevation).

---

**NOTE** TIN volume surfaces do not support editing operations.

---

The following illustration shows a surface with flat areas where the triangles (in cyan) of a surface are created from red contour lines. Some of the flat triangles and flat edges are indicated:
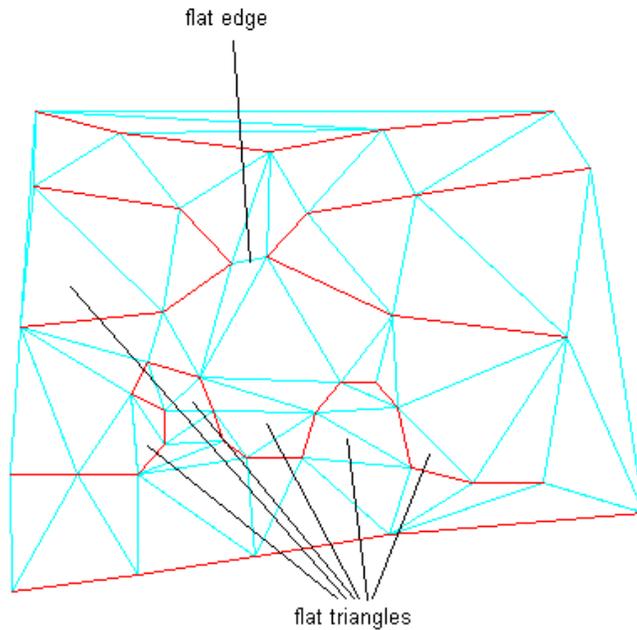


These flat areas represent inaccuracies in the surface. When the contours of such a surface are displayed, they might not match the original contour data around the flat areas. Also, displayed contours might contain points with three contour segments emanating from them instead of the expected two segments. Use the Minimize Flat Areas operation to find and eliminate triangles that have three points at the same elevation and edges that connect points on different data contours at the same elevation.

This command reduces the number of flat areas that result from the addition of contour data to a surface. It helps create a better surface model that realistically reflects a real-world surface.

The Minimize Flat Areas operation finds flat areas defined by triangle edges that are created from either a breakline or a contour. If a surface created from point data contains a flat area defined by three points with the same elevation, the operation does not affect the surface.

You can specify the Minimize Flat Areas By options when using the Minimize Flat Areas operation or during the Add Contour Data operation.

With one exception, the following Minimize Flat Areas options work only for TIN surfaces generated from contour data. In some cases, the Swapping Edges option can change the appearance of the contours displayed on a surface that is not generated from contour data.

---

**TIP** You can view flat triangles graphically by using slope shading. If you shade the surface using one range with a slope of 0.00 percent, you will see the flat areas along the contour data.

---

### Filling Gaps

This option fills the small gaps that sometimes occur in displayed contours and creates one continuous contour. If the two ends of the gap are close enough to be connected by a single triangle edge, the gap is filled by making that edge and the two data contours it joins into a single data contour. No additional points or edges are added.

If you want to control which gaps are filled, you can add contour data without selecting this option, and then use the Check for Contour Problems operation, which reports both ends of the gap. If, after reviewing the results of the operation, you identify gaps that you do not want filled, add a surface point between the two gap endpoints with an elevation above or below the contour. Then use the Minimize Flat Areas operation with the Filling Gaps In Contour Data option selected to fill the remaining gaps.

---

**NOTE** Gaps are filled in displayed contours only. The original surface contour data is not changed.

---

### Swapping Edges

This option scans the surface looking for a flat triangle that shares a non-contour edge with a non-flat triangle. If the two triangles form a convex quadrilateral, the common edge is swapped, creating two non-flat triangles. This may result in other flat triangles sharing non-contour edges with non-flat triangles, and these edges are swapped if possible. The process continues until no more swapping is possible.

This command reverses the direction of a TIN line between four adjacent surface points. To perform this operation, ensure that the surface style has points visibility enabled.

See Editing a Surface Style (page 694).

After the command is performed, the surface size remains unchanged and the surface contains the same number of points and triangles.

---

**IMPORTANT** The command does not remove all flat triangles and flat edges. Some of the new triangles may have extremely steep slopes.

---

flat edge

flat triangles

steep triangles (slope > 100%)
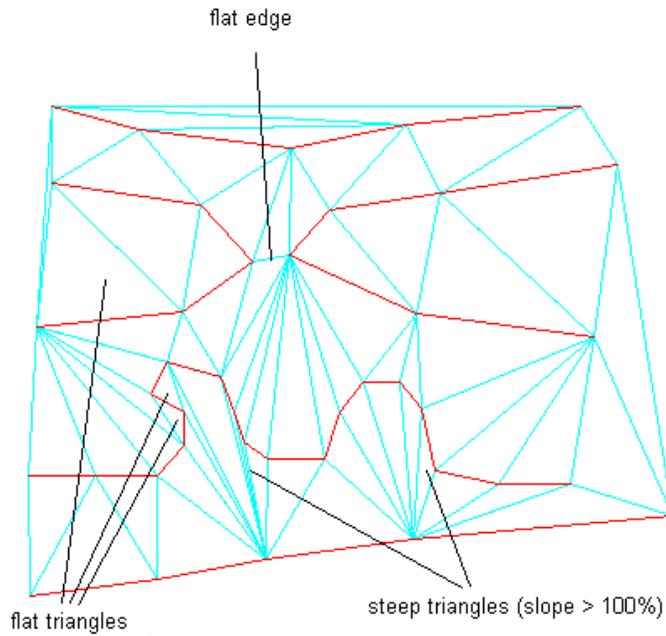
### Adding Points to Flat Triangle Edges

Like the previous option, this option scans the surface looking for a flat triangle that has all its points on data contours and shares a non-contour edge with a non-flat neighbor. However, instead of swapping the common edge of the triangle pair, a new point is added at the midpoint of the common edge. The elevation of the point is computed using Natural Neighbor Interpolation.

In many cases, this triangle falls at the edge of a chain of triangles that cross the flat area. Once you add the new point, the chain of flat triangles is traversed, with new points being added to flat triangle edges until the end of the chain is reached. The elevations of the points added along the chain are linearly interpolated between the elevations of the starting point and the ending point of the chain.

This option results in the removal of more flat triangles than the Swapping Edges option. Any triangle that would be removed by the Swapping Edges option is also removed by this option. This option also generally results in fewer steep triangles. However, additional points and triangles are added to the surface:
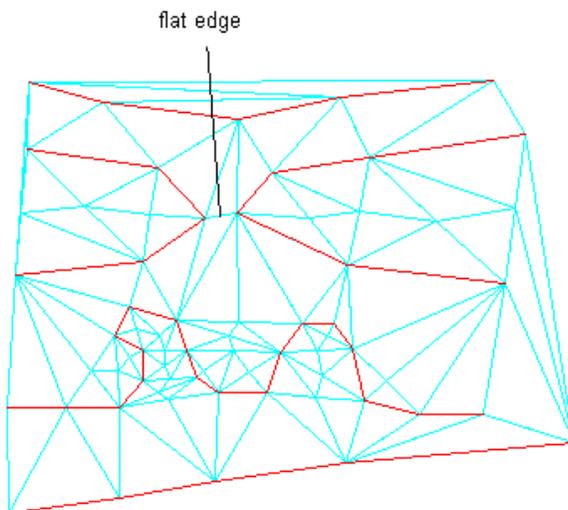


flat edge

In this illustration, 27 points are added to the surface. Most flat triangles are removed, except for some along the top and upper left borders.

**Adding Points to Flat Edges**

This option addresses flat edges, which are edges that bridge two same-elevation data contours but are not in flat triangles. The surface is scanned for any flat edges with both endpoints on data contours. If the opposing points of the two triangles containing such an edge are both at a higher elevation than the edge, or both at a lower elevation than the edge, then a new point is added at the midpoint of the flat edge. The elevation of the new point is computed using Natural Neighbor Interpolation. Because new points and edges are added, this option increases the size of the surface.

The following illustration shows the same surface after applying both the Adding Points To Flat Triangle Edges and the Adding Points To Flat Edges options. A total of 28 points are added in this example.
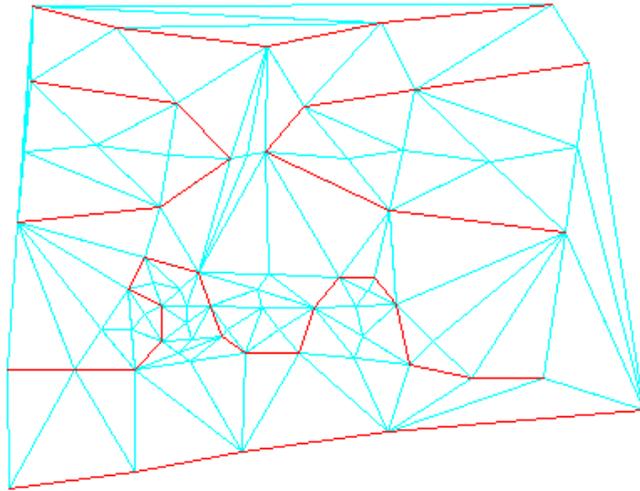


One benefit of the Swapping Edges option is that it does not increase the size of the surface. The Adding Points To Flat Triangle Edges and Adding Points To Flat Edges options generally produce better results, but they increase the size of the surface.

---

**TIP** You can differentiate between the display of points that are created from the contour data and the points that are added by the Minimize Flat Areas operation by changing the surface style. On the Points tab in the surface style, make the Data Point Symbol and Color different from the Derived Point Symbol and Color.

---

Before using the Minimize Flat Areas operation on a surface, you might find it helpful to evaluate flat areas so you can control the results of the operation, as shown in the following examples.

The following illustration shows a surface generated for a ridge where an open contour ends near the border of the surface but not on the border. The contours were added to the surface model with no Minimize Flat Areas. By options specified:

profile view of section line

section line

contour, elevation = 0          contour, elevation = 5
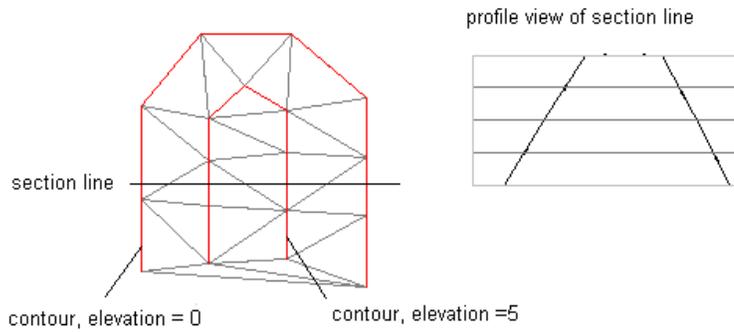
If you run the Minimize Flat Areas operation with the Adding Points to Flat Triangle Edges option, the following results are obtained:



profile view of section line

section line

smooth point created with
Natural Neighbor Interpolation

neighbor that causes smooth
point to have elevation < 10
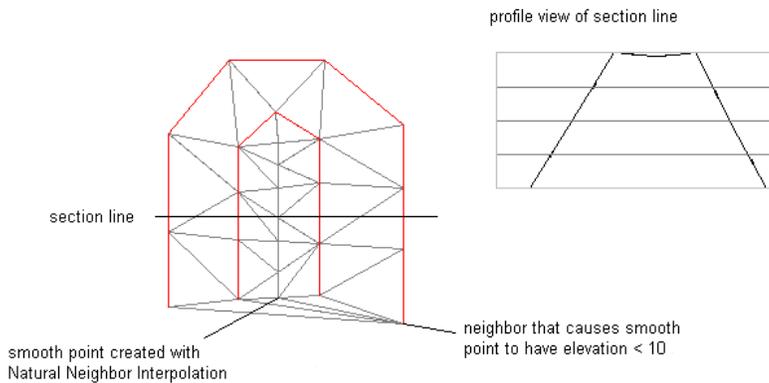
These results occur because the NNI interpolation uses surrounding points to determine an appropriate elevation for a new point. Because all surrounding elevations are lower than the flat triangle, elevations lower than the flat triangle are selected for the added points, resulting in the dip in the ridge.

In this instance, you can supplement the contour data by adding a spot elevation point near the open end of the contour, as shown in the following illustration. Then, the Minimize Flat Areas operation generates a crown as desired.



profile view of section line

section line

added point with elevation = 15

first smooth point created based
on NNI with new point has elevation > 10
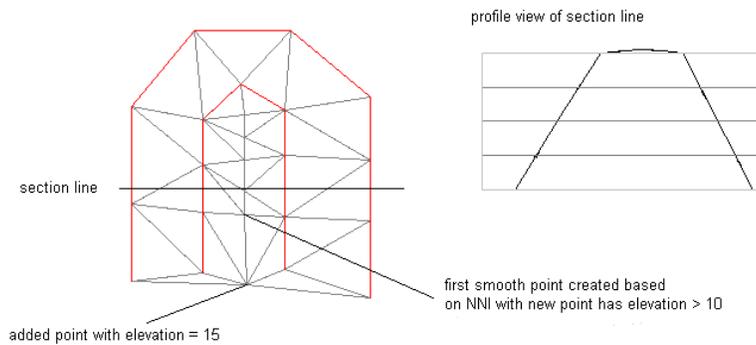
**NOTE** For best results, follow the suggested guidelines for the order of operations for creating contours and adding data to a surface. For more information, see Recommendations for Creating Surfaces from Contours (page 629).

**To minimize flat areas**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click , and click Minimize Flat Areas.

2   In the Minimize Flat Areas (page 2367) dialog box, select check boxes to specify Minimize Flat Areas By options.

    The selected options are performed in the order in which they appear in the dialog box.

3   Click OK.

    The operation is added as a Minimize Flat Areas operation to the Edits list view in the Prospector tree, with the selected options listed in the Description column of the list view.

### Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Minimize Flat Areas

Command Line

   MinimizeSurfaceFlatAreas

# Changing the Elevation of a Surface

Change the surface elevations by adding or subtracting an amount from the existing elevation of all the surface points, which raises or lowers the surface.

**NOTE** TIN or grid volume surfaces do not support editing operations.

A positive value increases the elevation; a negative value lowers the elevation.

**To raise or lower the surface**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click , and click Raise/Lower Surface.

    The following prompt is displayed:

    ```
    Amount to add to all elevations:
    ```

2   Enter a positive value to raise the surface or a negative value to lower the surface.

    The surface is adjusted to the new elevation.

    The operation is added as a Raise/Lower operation to the Edits list view in the Prospector tree.

    **NOTE** The Description column in the Prospector list view displays the value by which the surface was raised or lowered.

### Quick Reference

# Pasting Surfaces

Use the Prospector tab to paste a surface into the current surface.

**NOTE** TIN or grid volume surfaces do not support editing operations.

For example, if you have a building pad surface, you can paste it into a full-site surface. The building pad replaces that portion of the site surface that it covers.

**To paste a surface into the current surface**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click , and click Paste Surface.

2   In the Select Surface To Paste (page 1810) dialog box, in the list of surfaces, select the surface that you want to paste into the current surface, and click OK.

When the surface is pasted into the existing surface, the command adjusts the overlapping area to the elevation and contouring of the pasted surface.

The operation is added as a Paste Surface item to the Edits list view in Prospector.

**NOTE** The Description column in the Prospector list view displays the name of the surface that was pasted.

### Quick Reference

# Smoothing Surfaces

Surface smoothing adds points at system-determined elevations using Natural Neighbor Interpolation (NNI) or Kriging methods, which results in smoothed contours with no overlapping.

Only TIN surfaces support surface smoothing.

**NOTE** TIN volume surfaces do not support editing operations.

Surface smoothing resolves a problem associated with individual contour smoothing where the smoothing is applied to individual contour lines without regard to adjacent contours, which sometimes creates overlapping contours. Surface smoothing results in smoothed contours with no overlaps:



contours before smoothing          contours after smoothing
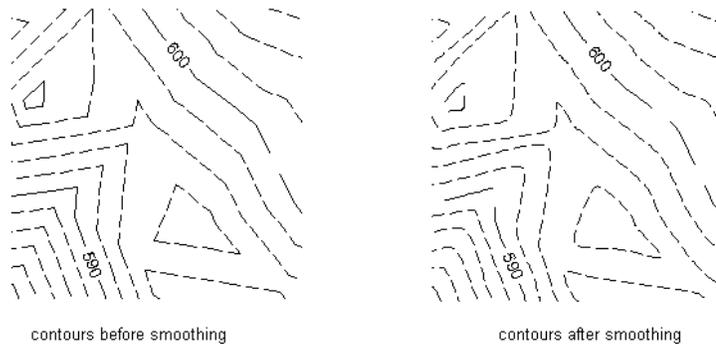
There are other benefits to surface smoothing. The Kriging method enables you to extrapolate, beyond the extents of a surface, based on the statistical trends across the existing surface. For example, if a site contains a random sample of borehole elevations (a sparse set of data points), you can statistically extrapolate a representation of a surface.

Because smoothing is an edit operation performed on a surface, you can specify smoothing properties and turn them on or off. When you turn the smoothing off, the surface reverts back to its original state, but the smoothing properties are preserved in the surface operation list. For more information about the operation list, see Understanding the Surface Definition (page 611).

**NOTE** Surface smoothing can take a considerable amount of time proportional to the number of output points. For the Kriging method, it is recommended that you use a relatively small sample point set (no more than 100 to 200 points).

## Understanding Point Interpolation/Extrapolation Output

Choose between several options when you use either of the NNI or Kriging surface smoothing methods for the interpolation and extrapolation of the point output.

**NOTE** For more information about the parameters, see the Smooth Surface (page 2357) dialog box help.

### Grid Based Location

Use the Grid Based output location to interpolate surface points (NNI), or interpolate/extrapolate surface points (Kriging) on a grid defined within specified polygon areas selected in the drawing. After the areas are defined, the grid X and Y spacing, and orientation properties can be specified.

### Centroids Location

Use the Centroids output location to interpolate surface points (NNI and Kriging) at the existing surface triangle centroids within specified polygon areas selected in the drawing.

When this option is selected, the grid X and Y spacing and the orientation properties are disabled.

### Random Points Location

Use the Random Points output location to interpolate (NNI), or interpolate/extrapolate (Kriging) a specified number of random points within polygon areas selected in the drawing.

When this option is selected, the grid X and Y spacing and the orientation properties are disabled.

## Smoothing a Surface Using the Natural Neighbor Interpolation (NNI) Method

Use Natural Neighbor Interpolation (NNI) to estimate the elevation (Z) of an arbitrary point (p) from a set of points with known elevations.

The NNI method uses information in the triangulation from the known points to compute a weighted average of the elevations of the natural neighbors of point p. The number of neighbors (the number of points whose Z-values are averaged to get the interpolated value) depends on the triangulation: it is the number of points to which a new point (p) would be connected if inserted into the surface:
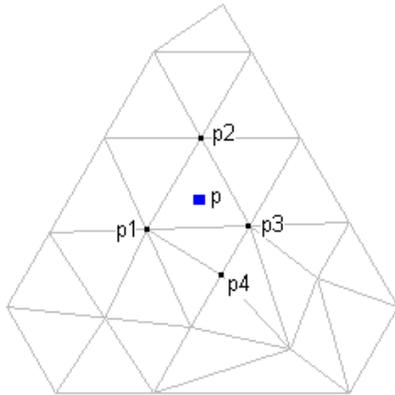


Using NNI, you select only the output locations of the interpolated points. The elevations of the interpolated points are always based on the weighted average of the elevations of the existing neighboring points. The outcome of the NNI method is more predictable than the Kriging method. NNI interpolates only within the surface, whereas Kriging can extrapolate beyond the surface border based on a selected polygon.

**To smooth a surface using Natural Neighbor Interpolation**

1   In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click  , and click Smooth Surface.

2   In the Smooth Surface (page 2357) dialog box, for the Select Method property, select Natural Neighbor Interpolation.

3   In Point Interpolation/Extrapolation Output, select the one of the Output locations: Grid Based, Centroids, or Random Points.

4   Depending on the location you selected, specify values:

  ■   **Grid Based**: Specify values for the output regions, grid spacing, and grid orientation properties.

  ■   **Centroids**: Specify the value for the output region.

  ■   **Random Points**: Specify the value for the output region.

5   Click OK to smooth the surface.

   The operation is added as a Smooth Surface item to the Edits list view in Prospector.

---

**NOTE**  The Description column in the list view displays the type of surface smoothing that was used (Kriging Smoothing or Natural Neighbor Smoothing).

---

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Smooth Surface

Command Line

  SmoothSurface

Dialog Box

  Smooth Surface (page 2357)

## Smoothing a Surface Using Kriging Method

Kriging is more complex than Natural Neighbor Interpolation. It requires both a model of the spatial continuity or dependence (in the form of a covariance or semivariogram), and a sample of surface data to determine the statistical trend on which to base interpolated/extrapolated points.

Spatial prediction using Kriging involves two steps:

1   Model the covariance or semivariogram of the spatial process. This involves choosing both a mathematical form and the values of the associated parameters.

2   Use this dependence model in solving the Kriging system at a specified set of spatial points, resulting in predicted values and associated standard errors.

### Sample Data

You must select the output locations of the interpolated points. It is important to ensure that the sample data is appropriate for the interpolated point locations (the output). For example, do not select points on the opposite side of the surface to determine a trend for the interpolated/extrapolated points locations, as that trend may not be appropriate for the interpolated/extrapolated point locations.

### Semivariogram Model

Semivariance is a measure of the degree of spatial dependence between samples. The magnitude of the semivariance between points depends on the distance between the points. A smaller distance yields a smaller semivariance and a larger distance results in a larger semivariance. The plot of the semivariances as a function of distance from a point is referred to as a semivariogram.

Kriging provides five semivariogram models:

■   **Exponential**

■   **Gaussian**

■   **Monomial**

■   **Linear** (default)

■   **Spherical**

The semivariance increases as the distance increases until at a certain distance away from a point the semivariance will equal the variance around the average value, and will therefore no longer increase, causing a flat region to occur on the semivariogram called a sill. The distance from the point of interest to where the flat region begins is termed the range or span of the regionalized variable. Within this range, locations are related to each other, and all known samples contained in this region, also referred to as the neighborhood, must be considered when estimating the unknown point of interest.

The center of the neighborhood is usually the unknown value. To determine this value, all known values within the neighborhood are assigned weights using the semivariogram. These weights and known values are then used to calculate the unknown value.

**TIP** It is recommended that you understand Kriging methodology before using it to perform surface smoothing.

**To smooth a surface using Kriging**

1 In Toolspace, on the Prospector tab, expand the surface Definition collection, right-click ✏ , and click Smooth Surface.

2 In the Smooth Surface (page 2357) dialog box, for Select Method property, select Kriging.

The Kriging Method properties are available and the Kriging semivariogram model is displayed at the bottom of the dialog box.

3 In Semivariogram Model, select the model.

4 Select a point selection method from the Point Selection Method list and following the prompts as required.

5 Depending on the semivariogram model that you select, you can specify values for Parameter A, Parameter C, and the Nugget Effect.

6 In Point Interpolation/Extrapolation Output, select the output locations: Grid Based, Centroids, or Random Points.

7 Depending on the location you selected, specify values:

■ **Grid Based**: Specify values for the output regions, grid spacing, and grid orientation properties.

■ **Centroids**: Specify the value for the output region.

■ **Random Points**: Specify the value for the output region.

8 Click OK to smooth the surface.

The operation is added as a Smooth Surface item to the Edits list view in Prospector.

**NOTE** The Description column in the list view displays the type of surface smoothing that was used.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Definition ➤ right-click Edits ➤ Smooth Surface

Command Line

SmoothSurface

Dialog Box

Smooth Surface (page 2357)

# Simplifying Surfaces

When you simplify a surface, you reduce the number of points or triangles in a TIN surface, while preserving the surface accuracy.

You can remove the specified amount of points in the specified surface region. The command does not remove points on breaklines and on the surface border.

**To simplify a surface**

1  Select a surface. On the Surface tab, Modify panel, click Edit Surface drop-down ➤ Simplify Surface ⬡ .

2  If more than one surface is available in the drawing, click the surface that you want to simplify or press Enter to select a surface in the Select A Surface dialog box.

3  Follow the Simplify Surface wizard sequence and the command line prompts to specify:

   ■  A simplification method (page 2371).

   ■  The surface region (page 2371) that you want to simplify.

   ■  Point reduction options (page 2371).

4  Click Apply.

   The total number of points to be removed is displayed.

5  Click Finish to remove the points.

The Simplify Surface operation appears in the surface definition in the Prospector tree.

## Quick Reference

Ribbon

  Select a surface. On the Surface tab, Modify panel, click Edit Surface drop-down ➤ Simplify Surface ⬡ .

Menu

  Surfaces menu ➤ Edit Surface ➤ Simplify Surface

Command Line

  SimplifySurface

Dialog Box

  Simplify Surface wizard (page 2370)

# Removing Surface Data

You can use several methods for removing data from a drawing, surface definition, or from both.

**NOTE** When deleting AutoCAD objects, the surface definition is deleted along with the AutoCAD object if the Copy Deleted Dependent Objects option in the surface definition is set to No. For more information, see Definition Tab (Surface Properties Dialog Box) (page 2346).

# Excluding Data from the Surface Build

Hide or disable operations in the current surface build while preserving them in the surface definition list.

Excluding data is useful for comparing scenarios. For example, you can add additional points to a surface from a point file, and later disable the Add Point File operation. The surface will revert back to the original (pre-point file) triangulation.

**To exclude data from the surface build**

**1** In Toolspace, on the Prospector tab, under the Surfaces collection, right-click the surface and click Properties.

**2** In the Surface Properties dialog box, click the Definition tab (page 2346).

**3** In the Operations List, clear the operations that you want to exclude from the build and click OK.

You are prompted with a warning to confirm the changes that you made to the surface definition.

**4** Click Yes to confirm the changes.

The surface is rebuilt with the unchecked operations excluded.

# Deleting Data from the Surface

Remove a data operation permanently while retaining the AutoCAD drawing objects (for example, polylines or points) on which the data operation may be based.

There are two methods to remove a data operation from a surface:

■ Delete it from the Prospector list view.

■ Remove it from the surface operations list in the Surface Properties dialog box.

**To delete data from the surface definition using the Prospector list view**

**1** In Toolspace, on the Prospector tab, expand the surface Definition collection and click the collection from which you want to delete data.

**2** In the Prospector list view, right-click the list item and click Delete.

**3** Click Yes to confirm the deletion of the selected breaklines.

The surface is rebuilt without the deleted operation.

**To delete data from the surface operations list**

**1** In Toolspace, on the Prospector tab, right-click the surface and click Properties.

**2** In the Surface Properties dialog box, click the Definition tab (page 2346).

**3** In the Operations List, right-click the operations that you want to delete from the surface and click Remove From Definition.

You are prompted with a warning to confirm the changes that you made to the surface definition.

**4** Click Yes to confirm the changes

The surface is rebuilt with the deleted operations excluded.

# Deleting Data from the Surface and the Drawing

You can permanently remove both a surface definition item and the AutoCAD object from which it is derived.

**NOTE** If you delete an AutoCAD object, the surface definition item is deleted only if the Copy Deleted Dependent Objects option in the surface definition is set to No. For more information, see Definition Tab (Surface Properties Dialog Box) (page 2346).

**To graphically delete data from the surface and drawing**

■ In the drawing area, select the object and press Delete.
   Both the object and surface definition item are deleted.

# Deleting AutoCAD Objects

You can permanently remove the AutoCAD objects that surface definition items are derived from without deleting the surface definition item.

For all surface definition items that are derived from AutoCAD objects, ensure that the surface build parameter Copy Deleted Dependent Objects is set to Yes. Then you can graphically select and delete the AutoCAD object while preserving the surface definition.

Also, you can use the Prospector list view to delete the AutoCAD objects from which the boundaries and breaklines are derived.

**To delete AutoCAD objects from the drawing graphically**

■ In the drawing area, select the object and press Delete.
   The object is deleted.

> **NOTE** If you want to delete an AutoCAD object, and not delete the surface definition item ensure that the Copy Deleted Dependent Objects option in the surface definition is set to Yes. For more information, see Editing and Viewing the Surface Definition (page 681).

# Masks

Use masks to block out areas of a surface and prevent them from being displayed or to render a section of the surface using a specified render material.

Masks can be defined from AutoCAD Civil parcels and from the following types of polygon-type objects:

■ Polylines                    ■ Faces

■ 3D Polylines                 ■ Survey Figures

■ Circles                      ■ Feature Lines

■ Ellipses                     ■ Parcel Segments

■ Rectangles                   ■ Surfaces

When you define a mask from a selected object, the mask geometry automatically updates when you make changes to the original object. If the original object is deleted from the drawing, the mask is also deleted.

The display of the mask is based on the display settings of the referenced surface style. The display type is either trimmed to the segments of the mask or crosses the segments:

| Display Type | Trim/Cross |
| --- | --- |
| Points | Trim |
| Triangles | Cross |
| Border | N/A* |
| Contours | Trim |

| Display Type | Trim/Cross |
|---|---|
| Grid | Trim |
| Directions | Cross |
| Elevations | Cross |
| Slopes | Cross |
| Slope arrows | Cross |
| Watersheds | Cross |

**\*** Border display is not affected by masks.

The following illustration is an example of crossing display (triangles):
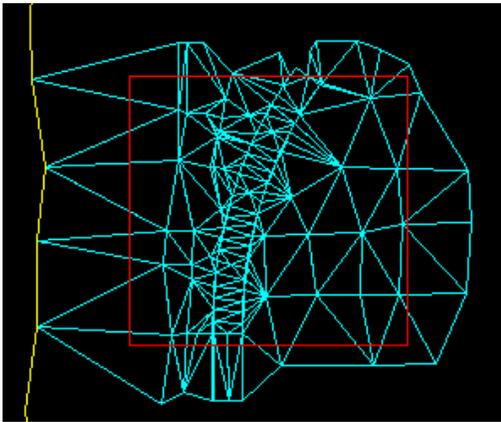


The following illustration is an example of trimmed display (contours):
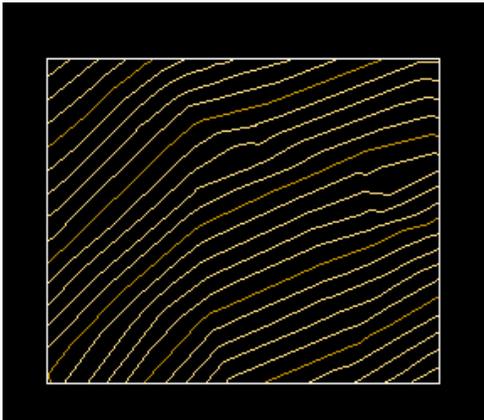


# Types of Masks

Define one of the two types of surface masks: outside or inside.

**NOTE** If you define an inside mask, you cannot use an outside mask on the same surface.

**Outside Masks**

When you define a mask as an outside type, the portion of the surface outside the defined polygon is hidden (the outside display is clipped to the mask segments):
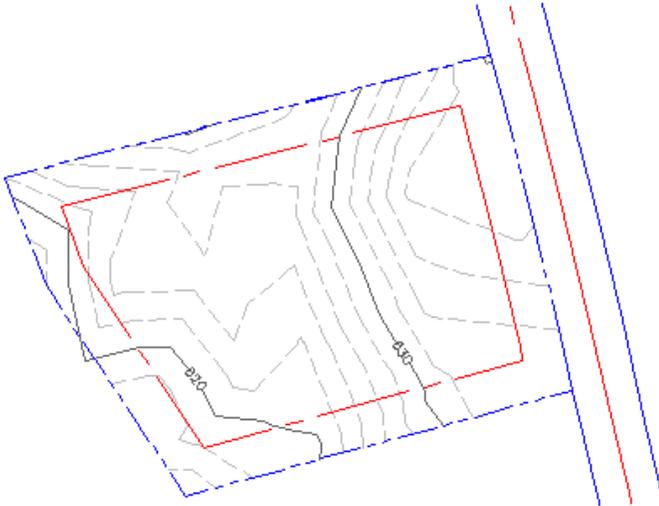


**Inside Masks**

When you define a mask as an inside type, the portion of the surface residing inside the defined polygon is hidden:
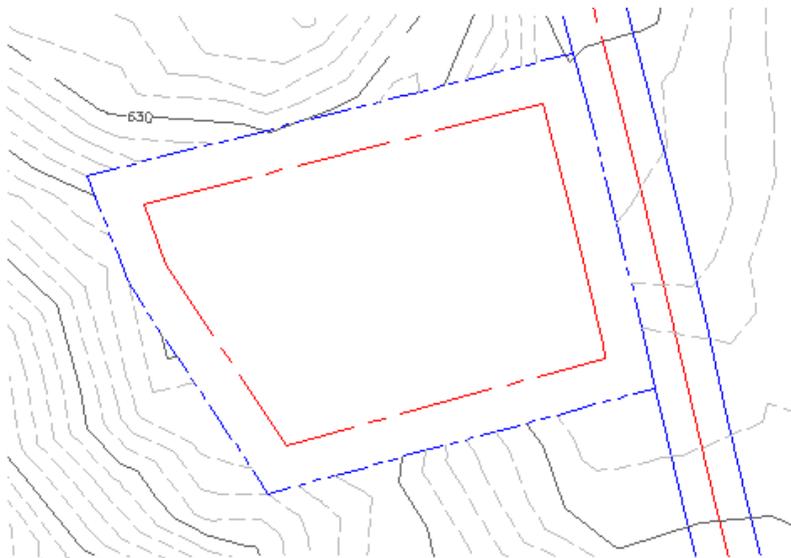


**NOTE** Only one inside mask can be defined on a surface.

**Mask Render Only Property**

When you define an inside or outside mask as render only, the specified render material is applied to the inside or the outside of the mask polygon. The display of the surface is not masked, but when the surface is rendered, the render material pattern is displayed. The following illustration shows the render material applied to an inside mask:

# Creating a Mask

Create masks by selecting existing parcels or AutoCAD polygon-type objects.

**To create a mask**

1   In Toolspace, on the Prospector tab, expand the surface collection, right-click 🏠 Masks, and click Create Mask.

2   From the drawing, select an object or multiple objects to define the mask. Press *Enter.*

3   In the Create Mask (page 2363) dialog box, in the properties grid, click the Value column for the Name property and enter a name for the mask.

4   Click the Value column for the Description property and enter a description for the mask.

5   For Mask Type property, select Inside or Outside.

> **NOTE** If you define an inside mask on a surface, you cannot create another inside or outside mask on the same surface.

6   Depending on whether or not you want to set a selected object as a render-only mask, select Yes or No for Render Only option.

> **NOTE** If multiple objects are selected for the mask, the Render Only option is set to true and is unavailable.

7   Optionally, if the polygon object from which you are creating a mask has curves, enter a value in the Mid-Ordinate Distance field or click 📐 to digitize a distance in the drawing area.

8   Specify the render material by clicking in the Render Material row and clicking ⋯ .
The Render Material dialog box is displayed. For more information, see Applying Render Materials to Objects (page 1578).

9   Click OK.
The mask is created and added to the surface Masks collection in the Prospector tree.

**NOTE** A render-only mask must be rendered before it is displayed. See Rendering Objects (page 1579)

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ right-click Masks ➤ Create Mask

Command Line

CreateSurfaceMask

Dialog Box

Create Mask (page 2363)

# Viewing Masks

Select and view masks in the Prospector tab item view.

In the Prospector item view, you can manipulate the view of the mask, including zooming, panning, rotating, and orbiting. You can also save the view to an image file.

A render-only mask must be rendered before you can view it. For information, see Rendering Objects (page 1579).

### To view a mask in the Prospector item view

1 In Toolspace, on the Prospector tab, expand the surface ⌂ Masks collection and click the mask's name. The mask is displayed in the Prospector item view.

2 Right-click anywhere in the item view and select the option you want, such as Zoom To or Pan To.

# Changing the Display Order of Render Only Masks

Use the Mask Display Order dialog box to change the display order of a selected render-only mask.

A mask with a higher display priority is drawn last and on top of any overlapping masks.

### To change the display order of a render-only mask

1 In Toolspace, on the Prospector tab, right-click the surface ⌂ Masks collection and click Display Order. The Mask Display Order (page 2363) dialog box is displayed.

2 From the list, click a mask and click ⬆ to move the mask up in the list or click ⬇ to move the mask down.
The mask at the top of the list has the highest display priority and is drawn on top of any overlapping masks.

3 Click OK to apply the changes and close the Mask Display Order dialog box.

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ right-click Masks ➤ Display Order

Dialog Box

Mask Display Order (page 2363)

# Modifying a Mask

Use the Mask Properties dialog box to edit a mask's properties. You can also modify the extents, location, and positioning of a mask using the appropriate AutoCAD command (depending on the type of the originating object).

**To view and modify a mask's properties**

**1** In Toolspace, on the Prospector tab, click the surface ⌂ Masks collection.
The names and descriptions for the masks are displayed in the Prospector list view.

**2** Right-click the mask in the Prospector tree or list view and click Mask Properties.
The Mask Properties (page 2363) dialog box is displayed. Edit the name, description, boundary type, render material, and mid-ordinate distance.

**3** Click OK to apply the changes and close the Mask Properties dialog box.

**NOTE** You can move masks based on AutoCAD objects. You cannot move parcel-based masks.

**To move a mask using the Move command**

**1** At the command line, enter **move**.

**2** Select the mask you want to move.

**3** Follow the command line prompts.

**To modify (rotate, scale, or stretch) a mask**

■ Use the AutoCAD grip editing capabilities of the original objects (for example, polylines, and circles, and parcels) to modify masks.
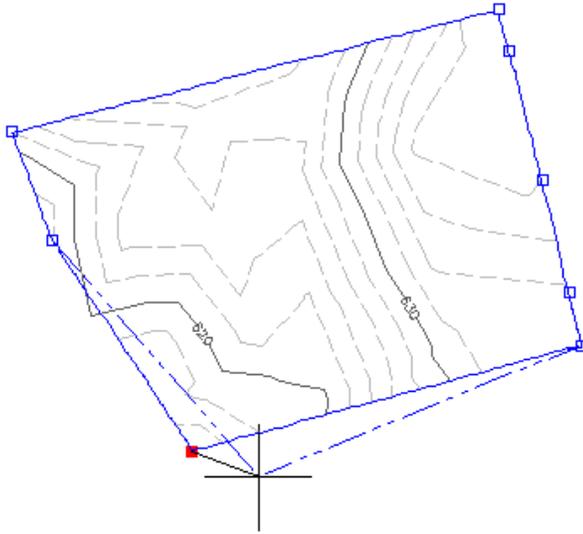
The mask reacts to changes made to the original object.

# Deleting a Mask

Use the Prospector tab to remove a mask from a surface by deleting the mask or by deleting the AutoCAD object that it was created from.

**To delete a mask**

1   In Toolspace, on the Prospector tab, click the surface Masks collection.

2   In the list of masks, right-click the mask and click Delete.

The mask is deleted from the surface, but the AutoCAD object (for example, a rectangle) is left in the drawing.

3   To delete the AutoCAD object, select it in the drawing area and press Delete.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ <surface-name> ➤ Masks ➤ right-click <mask-name> in list view ➤ Delete

# Watersheds

Use watersheds to analyze how water flows along and off of a surface.

Surface TIN lines are used to calculate the areas that water would flow along the surface. From these areas, the drain targets and watersheds are determined.

Different drain targets are calculated for watersheds. For example, some watersheds include a boundary point as a drain target, which is the point where the channel of water would drain off the surface. Other watersheds have depression areas that the water flows into, while some watersheds are defined as boundary segments where the drain target includes the boundary of a surface. For more information, see Types of Watersheds (page 676).

If AutoCAD Civil determines that water from one surface triangle could flow into more than one watershed, then it splits the TIN triangle to make two triangles. This ensures that each watershed consists of complete triangles, and that the boundary of each watershed consists solely of TIN edges.

# Types of Watersheds

Watershed types are based on the type of drain target the watershed has.

A drain target is the location where the water flow either stops or leaves the surface. Water that flows along an area or across a surface triangle eventually flows off the surface, or it reaches a point from which there is no downhill direction.

For each drain target in a surface, AutoCAD Civil determines the region of the surface that drains to that target. This region is called the watershed for that drain target.

Each watershed that you delineate is categorized as one of the following types, based on drain target.

## Boundary Point

If the downhill end of a channel edge is on the surface boundary, then water flowing through that channel continues off the surface. The boundary point is the lowest end of the channel.

If a watershed has this type of drain target, it is called a boundary point watershed.

In the following illustration, point p5 is a boundary point, the drain target of the channel p7-p6-p5:
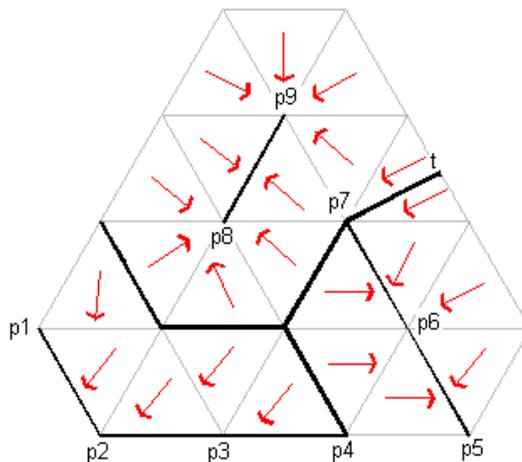


**NOTE** The "t" in the illustration indicates that the triangle was split between two watersheds because the water flowing across that triangle could go to either of two watersheds.

## Boundary Segment

If an edge on the surface boundary belongs to a triangle that slopes down toward that edge, then water flows off the surface all along that edge. A boundary segment is a connected sequence of such edges.

If a watershed has this type of drain target, it is called a boundary segment watershed.

In the following illustration, the edges p1-p2, p2-p3, and p3-p4 form a boundary segment.
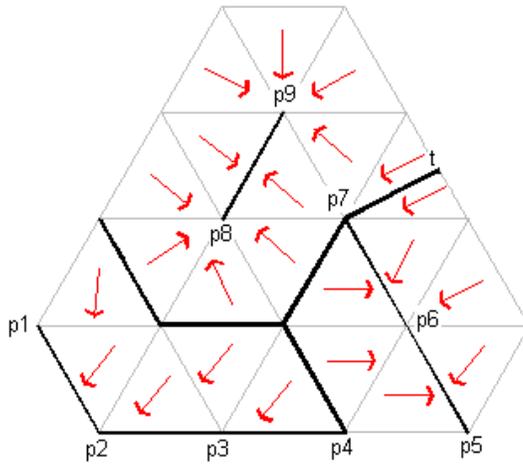
**NOTE** The "t" in the illustration indicates that the triangle was split between two watersheds because the water flowing across that triangle could go to either of two watersheds.

## Depression

If a point is at a lower elevation than all its neighboring TIN points, then when water flows to it, it has no downhill place to go. Similarly, a connected set of points that are at the same elevation and all of whose neighbors are at a higher elevation, is a single drain target. A depression is any such set of points.

If a watershed has this type of drain target, it is called a depression watershed.

In the following illustration, points p8 and p9 form a depression:
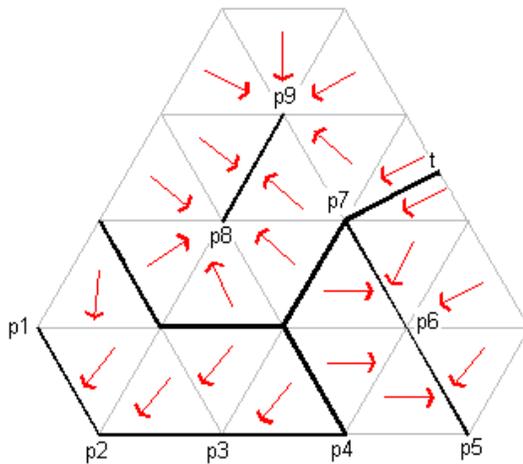


**NOTE** The "t" in the illustration indicates that the triangle was split between two watersheds because the water flowing across that triangle could go to either of two watersheds.

### Ambiguous Depression

A depression watershed can be ambiguous. A watershed is ambiguous if it does not include any drain targets, but water flowing on it can reach more than one drain target.

An ambiguous depression occurs when the depression watershed depth is less than the threshold depth, but there are multiple neighboring watersheds at points of minimum elevation on the boundary.

When such a watershed fills to overflowing, water flows to all those neighboring watersheds, so it cannot be merged into any single watershed. Instead, AutoCAD Civil keeps it as a separate watershed and lists the neighboring watersheds into which it will drain.

**Shallow Depression**

If a point is at a lower elevation than all its neighboring TIN points, then water that flows to it has no downhill place to go. Similarly, a connected set of points that are at the same elevation, and all of whose neighbors are at higher elevation, is a single drain target. A depression is any such set of points.

A shallow depression is determined by the average depth of a depression, which is the average depth of the depression when it is filled to overflowing. This is determined by dividing the value of water in the depression by the area of the top surface of the water. A depression is considered part of the watershed it drains into (provided there is only one watershed it drains into) if its average depth is less than the user-set threshold.

# Flat Area

A flat area watershed is a flat area, from which water could flow down to more than one drain target. It also includes the parts of the surface that drain to that flat area. A flat area is a connected set of triangles all of whose vertices have the same elevation. Flat areas abut parts of the surface that slopes downhill.

If for every edge on the boundary of a flat area, the opposing, non-flat triangle slopes up from the edge, then the flat area is the bottom of a depression watershed:
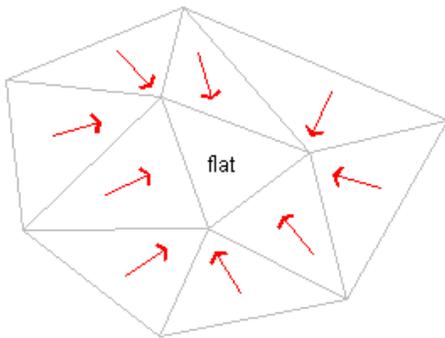


If some of the opposing, non-flat triangles slope down from the flat area boundary, but all flow to the same drain target, then the flat area is part of the watershed for that drain target "a", as shown in the following illustration:
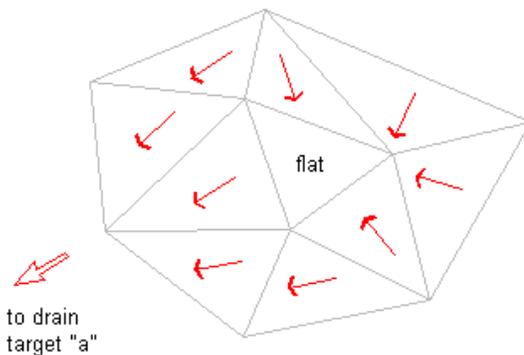


In the following illustration, the flat area, plus whatever part of the surface flows down to it, becomes a flat area watershed. This watershed is ambiguous because water flowing through it can flow to more than one drain target, "a" and "b":
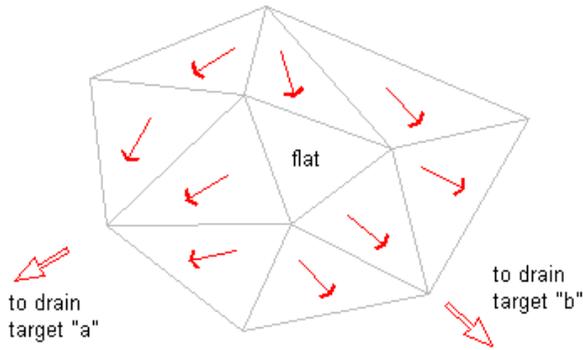
## Multi-Drain

One type of ambiguous watershed is called a multi-drain or split channel watershed.

In the following illustration, the channel edges e2 and e3 flow to different drain targets:

Water flowing down edge e1 could eventually reach either of these drain targets. In a case like this, AutoCAD Civil determines the region that flows to edge e1 and defines this region to be a multi-drain watershed.

Also, in this situation, AutoCAD Civil keeps track of the watersheds into which water from the multi-drain watershed might drain, in this case "a" and "b", as shown in the following illustration:
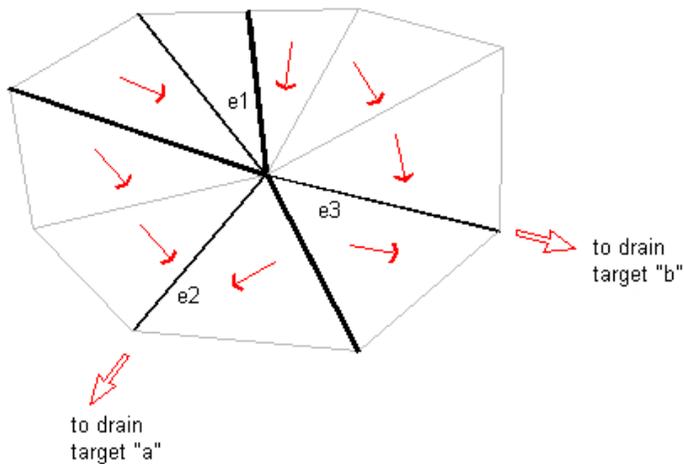


## Multi-Drain Notch

This type of watershed is called a multi-drain notch because water flowing into the notch formed from p1 and p2 and could drain to drain target "a" or drain target "b" as is shown in the following illustration:

A multi-drain notch watershed occurs where there is a a flat edge between two points on a surface:
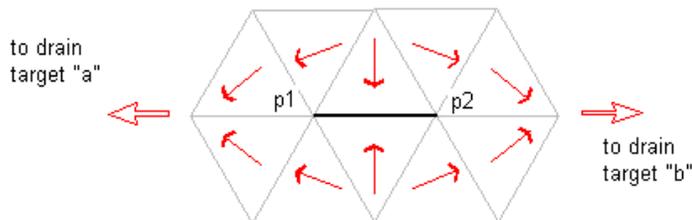
# Generating and Displaying Watersheds

Generating and displaying watersheds on a surface is a flexible process.

**To generate and display watersheds**

1 Configure the watershed display or style settings. For information, see Creating a Surface Style (page 693).

2 Configure the watershed analysis and legend. For information, see Analyzing Surfaces (page 708).

3 Display the watershed. Watershed display is set on the Surface Style Display (page 2345)and Surface Properties Analysis (page 2343)tabs.

4 Insert the watershed legend table. For information, see Adding Surface Legend Tables (page 707).

# Viewing Watershed Properties

After you have created a watershed analysis on a surface, the watersheds are added to the surface Watersheds collection in the Prospector tree.

For information about creating a watershed analysis, see Analyzing Surfaces (page 708).

**To view watershed properties**

1 In Toolspace, on the Prospector tab, click the surface Watersheds collection.

The Prospector list view displays a tabular list of the surface watersheds with their IDs, description, type, and the ID of the watershed that they drain into.

2 Optionally, to pan or zoom to an individual watershed, right-click the watershed item in the list view and click Pan To or Zoom To.

> **NOTE** The watershed display needs to be enabled before you can view it. For information on displaying watersheds, see Surface Styles and Visualization (page 692).

# Managing Surfaces

You can manage instances of surfaces in the Surfaces collection in the Prospector tab and surface settings in the Surface collection in the Settings tab.

You can copy, export to LandXML, delete, rebuild, rename, synchronize, take a snapshot of, and change the style usage for a surface.

Every surface object has properties, which are defined and displayed in the Surface Properties dialog box (page 2346).

Surface properties include:

■ General information, including the name, description, and style for the surface. For information, see Information Tab (Surface Properties Dialog Box) (page 2346).

■ Definition attributes, including the build, data, and operation options, as well as a definition of items that are in use in the surface. For information, see Understanding the Surface Definition (page 611).

■ Analysis information, which includes properties for all types of surface analysis, including contour, direction, elevations, slopes, slope arrows, and watersheds. For information, see Analyzing Surfaces (page 708).

■ Statistics for the current state of the surface. For information, see Viewing Surface Statistics (page 689).

# Editing and Viewing the Surface Definition

Edit and view a surface definition information, including build, data, and operation options. Additionally, you can select or clear items that are included in the surface definition list.

For more information, see Understanding the Surface Definition (page 611).

**To edit the surface build and definition information**

1   In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Properties.

2   In the Surface Properties dialog box, click the Definition tab (page 2346).

3   To edit the general build information, expand the Build group and edit the values.

4   To specify whether certain data categories (for example, breaklines) are to be excluded from or included in the surface build, expand the Data Operations definition group and set the corresponding values to Yes or No.

5   To specify whether certain edit operation categories (for example, deleting points) are to be excluded from or included in the surface build, expand the Edit Operations group and set the corresponding values to Yes or No.

6   To reorder any of the surface operations, click the arrow buttons to the left of the Operation Type field.

7   Click Apply to make the changes or click OK to make the changes and close the Surface Properties dialog box.

    A warning message, which enables you to confirm or cancel your changes, is displayed.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Properties

Object Shortcut Menu

Surface Properties

Command Line

EditSurfaceProperties

Dialog Box

Surface Properties - Definition Tab (page 2346)

# Using Surface Snapshots

A snapshot is a surface operation that captures the current state of surface.

When a surface snapshot exists and the surface is rebuilt, the build process starts at the snapshot and proceeds sequentially through subsequent surface operations. Therefore, if a surface operation, such as an import of an external data file, exists prior to the snapshot operation, the surface build process does not attempt to find and open the external data file since all of the external file point data is contained in the surface snapshot operation.

## Creating a Snapshot

Use the Prospector tab to create a surface snapshot.

The current surface points and triangles resultant from previous surface operations are contained in the surface snapshot operation. When a surface is built, previous surface operations are ignored and the surface build begins at the snapshot operation.

Snapshots can be created automatically. A LandXML setting enables you to specify that a surface snapshot be created in the surface definition subsequent to the importing of a surface when using Import LandXML. For more information, see Viewing and Editing the LandXML Import Settings (page 1678).

**To create a snapshot**

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Create Snapshot.

---

**NOTE** If there is an existing snapshot operation, a warning dialog box is displayed from which you can confirm or cancel the operation.

---

Surface operations that are included in the snapshot are marked with  in the Prospector tree.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click <surface-name> ➤ Create Snapshot

## Rebuilding a Snapshot

You can rebuild a snapshot if any of the operations that precede it in the surface operation list change or are removed.

If the snapshot operation is out-of-date,  is displayed next to its node in the Prospector tree.

**To rebuild a snapshot**

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Rebuild Snapshot.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click <surface-name> ➤ Rebuild Snapshot

## Deleting a Snapshot

You can remove an existing snapshot.

**To delete a snapshot**

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Remove Snapshot.

■ A warning dialog box is displayed in which you can confirm or cancel the operation.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click <surface-name> ➤ Remove Snapshot

## Snapshots and the Surface State

The out-of-date icon can be displayed at several places in the Prospector tree or surface definition depending on the status of the snapshot and the surface.

■ Out-of-date snapshot. When an active snapshot is out of date, 🔰 is displayed next to the snapshot operation in the Definition tab of the Surface Properties dialog box.

■ Out-of-date surface. If the surface is out of date when you attempt to create a snapshot, a warning dialog box is displayed, indicating that the snapshot will be created based on the current (out-of-date) state of the surface.

■ Out-of-date data. If the surface data is out of date (Rebuild - Automatic is disabled) and is contained in an active snapshot, the data item is shown as out of date 🔰 in the Prospector tree, but the surface is not shown as out of date.

# Changing a Surface Name, Description, or Style

Change an existing surface name, description, and style usage.

**To rename a surface**

1 In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Properties.

2 In the Surface Properties dialog box, click the Information tab (page 2346).

3 To edit the name of the surface, enter a new name in the Name field.

4 To edit the description of the surface, enter a new description in the Description field.

5 To change the style for the surface, select a style from the Object Style list or use the standard style selection controls.

6 Click Apply to make the changes, or click OK to make the changes and close the Surface Properties dialog box.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Properties or Prospector tab: Surfaces ➤ click surface <property-name> in list view

Object Shortcut Menu

Surface Properties

Command Line

EditSurfaceProperties

Dialog Box

Surface Properties - Information Tab (page 2346)

# Copying a Surface

Create a copy of an existing surface.

When a surface is copied, the copy is named using a naming convention. For example if the original surface that is copied is named 'Surface 1', the new surface is named 'Surface 1 (1)'. If Surface 1 is copied again, the new surface is named 'Surface1 (2)'. If 'Surface 1(1)' is copied, the new surface is named 'Surface 1 (1) (2)'.

**To copy a surface**

1   In the drawing area, select the surface, right-click, click Basic Modify Tools ➤ Copy.

2   At the command line, you are prompted to specify a base point for the surface.

3   Pick an insertion point in the drawing area.
    The surface is copied in the drawing and added as a new surface to the Prospector tree.

4   To edit the name of the surface, right-click the surface in the Prospector tree and click Properties.
    The Surface Properties dialog box is displayed with the Information tab (page 2346) active. The Name field contains the name of the copied surface, which is, by default " <surface-name> (1)."

5   Enter a new name in the Name field.

6   To edit the description of the surface, enter a new description in the Description field.

7   To change the style that the surface is using, select a style from the Object Style list or use the standard style selection controls.

8   Click Apply to make the changes, or click OK to make the changes and close the Surface Properties dialog box.

## Quick Reference

Object Shortcut Menu

Surface Properties

Dialog Box

Surface Properties - Information Tab (page 2346)

# Previewing a Surface

You can preview a surface in the Prospector item list view.

**To preview a surface**

1   In Toolspace, on the Prospector tab, right-click the Surfaces collection and click Show Preview.

    **NOTE** A check mark is displayed next to the Show Preview menu item when the preview is enabled.

**2** In the Prospector tree, click the surface that you want to preview.

The surface is displayed in the Prospector item view.

> **NOTE** The preview of the surface uses the same display styles as the surface in the drawing area.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click Surfaces ➤ Show Preview

## Rebuilding a Surface

When you make changes to a surface, such as editing data or the build parameters, the surface may become out of date and you can rebuild it.

If the surface definition is out of date, ▽ is displayed next to its node in the Prospector tree.

AutoCAD Civil enables you to either automatically or manually rebuild a surface.

### To automatically rebuild a surface

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface and click Rebuild - Automatic.

> **NOTE** A check mark is displayed next to the Rebuild - Automatic menu item when it is enabled.

When changes are made to the surface, it is automatically rebuilt and updated.

### To manually rebuild a surface

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Rebuild.

> **NOTE** If the surface definition is out-of-date, ▽ is displayed next to its node in the Prospector tree.

The surface is rebuilt and updated in the drawing.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Rebuild

## Deleting a Surface

Delete a surface, which erases it from the drawing and removes it from the Surfaces collection in the Prospector tree.

### To delete a surface

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Delete.

The surface is erased from the drawing and removed from the Surfaces collection in the Prospector tree.

**NOTE** You cannot delete a surface from the Prospector tree if it has associated dependencies (such as other objects). You can determine if the surface has dependencies by the orange triangle attached to the surface icon. The Delete command for such surface is missing from the shortcut menu.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Delete

## Locking a Surface

Lock a surface so that its properties and geometry cannot be changed. Certain tabs in the Surface Properties dialog box and some menu items cannot be selected.

For more information about controlling access to a project surface using AutoCAD Civil project management functionality, see Using Vault (page 138).

**To lock a surface**

■ In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Lock.

In the Prospector tree, '✍ is displayed next to the surface.

The surface is locked and its geometry cannot be edited.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Lock

## Moving, Scaling, or Rotating a Surface

Use standard AutoCAD commands to move, scale, and rotate a surface.

For more information about each AutoCAD command, refer to the Help.

**NOTE** The AutoCAD Mirror command is not supported on surfaces.

**To move, scale, or rotate a surface**

1 At the command line, enter the command, for example **Move**.

2 Follow the command-line prompts.

**NOTE** When moving, scaling, or rotating a surface, you should be aware that if you want to affect only the surface and not the data, you should select only the surface. After moving, scaling, or rotating as surface, a 'Transform By' operation is added to the surface definition and is reapplied during subsequent rebuilding of the surface. If the surface data in the drawing needs to be moved, scaled, or rotated, then the surface should not be selected since the surface is rebuilt according to location of its dependent data.

# Editing Surface Settings

Use surface settings to specify the default behavior for surface-related commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Settings tree. You can control settings at three levels: the drawing level, the object collection (feature) level, and the command level. It is important that you understand how the different levels of settings work together. For more information, see Specifying Drawing Settings (page 63).

You can use the Settings tree Surface collection shortcut menu to establish defaults for all surface-related commands. You can change surface-specific settings at this level, or you can override the drawing ambient settings.

Use the Commands collection under the Surface collection to change surface settings for a specific command. You can change surface-specific settings at this level, or you can override the drawing ambient settings.

**NOTE** Overrides to the drawing ambient settings at the Surface collection level and the surface Commands collection level affect only the specified level. The drawing level settings are not changed.

The topics in this section describe only those settings that affect surface-related commands, and do not cover the drawing ambient settings that you can change at the surface collection level and the surface command level, even though those settings are displayed in the Edit Feature Settings dialog box. For more information, see Specifying Ambient Settings (page 68).

## Changing Surface Creation Settings

Use the Surface Creation settings in the Edit Feature Settings - Surface (page 2329) dialog box to specify surface-related settings before you create surfaces.

For example, before creating or importing surfaces, you may want to specify the default surface style and default surface label styles.

If a closed lock appears in the Lock column for a property, the property is locked at a higher level of the Settings tree and cannot be changed at this level.

If you change a property value, a check mark appears in the Override column. This override is also reflected in the property table for related objects higher up the Settings tree, where an arrow appears in the Child Override column.

**To the change surface creation settings**

1  Do one of the following:

   ■ **To edit settings for all surface-related commands**: In Toolspace, on the Settings tab, right-click the Surface collection and click Edit Feature Settings.

   ■ **To edit settings for a specific command**: In Toolspace, on the Settings tab, expand the Commands collection under the Surface collection. Right-click the name of the command for which you want to change settings and click Edit Command Settings.

   The Edit Feature Settings - Surface (page 2329)dialog box is displayed.

2  Expand the Surface Creation property group.

3  To specify the default surface type, edit the Surface Default Type settings.

4  To specify the default grid surface spacing and orientation, edit the following settings:

   ■ Grid Surface X-Spacing

   ■ Grid Surface Y-Spacing

   ■ Grid Surface Orientation

**5** To specify the default name format for a surface, expand the Default Name Format property group and edit the Surface Name Template setting.

**6** Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Surface collection ➤ Edit Feature Settings
Or
Settings tab: Surface ➤ Commands ➤ right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Surface (page 2329)

# Changing Surface Style Settings

Use the Surface Style settings to specify default surface styles and surface label styles before you create surfaces.

You can use the default styles specified in these settings to establish the default setting for the style in the Create Surface dialog box.

**To change the surface style settings**

**1** Do one of the following:

- **To edit settings for all surface-related commands**: In Toolspace, on the Settings Tree, right-click the Surface collection and click Edit Feature Settings.

- **To edit settings for a specific command**: In Toolspace, on the Settings tree, expand the Commands collection under the Surface collection. Right-click the name of the command you want to change settings for and click Edit Command Settings.

**2** Expand the Default Styles property group.

**3** To specify a default surface style, change the Surface Default Style setting.

**4** To specify a default spot elevation label style, change the Surface Spot Elevation Label Style setting.

**5** To specify a default slope label style, change the Surface Slope Label Style setting.

**6** To specify a default point style, change the Point Style setting.

**7** Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Surface collection ➤ Edit Feature Settings
Or
Settings tab: Surface ➤ Commands ➤ right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Surface (page 2329)

# Viewing Surface Statistics

AutoCAD Civil provides extensive statistics based on the current state of the surface.

Surface statistics categories include:

- **General**. Common to all surface types. Includes information such as bounding extents and elevations.
- **Extended**. Available for (non-volume) TIN and grid surfaces. Includes information such as 2D and 3D area as well as grade and slope values.
- **Grid**. Specific to grid surfaces. Includes information such as grid spacing and orientation.
- **TIN**. Specific to TIN surfaces. Includes information the number, area, and length of surface triangles.
- **Volume**. Specific to volume surfaces. Includes volume statistics.

**To view the statistics for a surface**

1  In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Properties.

2  In the Surface Properties dialog box, click the Statistics tab (page 2352).

3  To view the statistics for the surface, expand the corresponding category.

## Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Properties

Object Shortcut Menu

   Surface Properties

Command Line

   EditSurfaceProperties

Dialog Box

   Surface Properties - Statistics Tab (page 2352)

# Troubleshooting Surface Errors

Identify and troubleshoot errors or problems with a surface, or its data components, in several ways.

For surface troubleshooting information, see Surface Errors and Issues (page 647).

# Working with Large Surfaces

In AutoCAD Civil you can build, edit, and save large surfaces. The term "large surface" applies to a surface with a number of points exceeding one million (generally, 1 million points for a grid surface or a snapshot of a TIN surface, and 2 million points for a TIN surface).

AutoCAD Civil has a mechanism for handling operations with large surfaces. When you save a drawing containing a large surface, the data pertaining to the large surface is written to a separate file that is saved in the same location where the surface drawing resides. The TIN surface companion files have the *.mms* file extension, and the grid surface files have the *.grs* file extension. The file name of a surface companion file has the *<drawing_name>_<surface object handle>.mms* format.

**NOTE** Every time you move the drawing file to a different location, you must move the companion file to the same location.

For more details on working with large surfaces, see:

# Calculating Surface Volumes

Use surface utilities to query composite and bounded volume differences between surface.

Composite volumes use top and bottom surfaces (a surface pair) to establish cut, fill, and net volume values. Bounded volumes use an existing AutoCAD object (for example, a polyline or polygon) to calculate the cut, fill, and net volume for the area bounded by the object.

# Calculating Composite Volumes

Calculate volumes using the composite method, which triangulates a new surface, based on points from both surfaces.

This method uses the points from both surfaces, as well as any location where the edges of the triangles between the two surfaces intersect to create prismoidal segments from composite TIN lines.

The new composite surface elevations are calculated based on the difference between the elevations of the two surfaces, as follows:
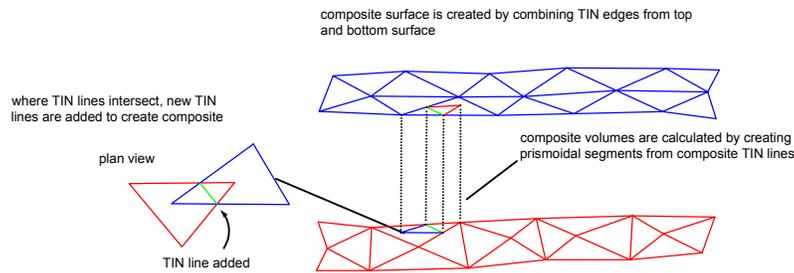


This method gives accurate volume measurements between the two surface definitions.

**See also:**

■ Creating a TIN Volume Surface (page 604)

■ Creating a Grid Volume Surface (page 606)

**To calculate composite surface volumes**

**1** Open, create, or import the TIN or grid surfaces for which you want to measure the composite volume. See Creating Surfaces (page 601).

**NOTE** The Composite Volumes utility compares two surfaces (surface pairs), so you must have two surfaces available in your drawing.

**2** Click Analyze tab ➤ Volumes And Materials panel ➤ Volumes drop-down ➤ Volumes 🗔 .

**3** Do one of the following:

■ In the Composite Volumes (page 2364) vista, click  and select the surfaces to compare by clicking the <select surface> entry from the Base Surface and Comparison Surface columns (for the base and comparison surfaces respectively).

■ In the Composite Volumes vista, click  and select both the base and comparison surfaces from the drawing area.

After you select the surfaces, the volumes are calculated and the following information is displayed:

■ **Cut**. The amount of material that has to be removed.

■ **Fill**. The amount of material that has to be added.
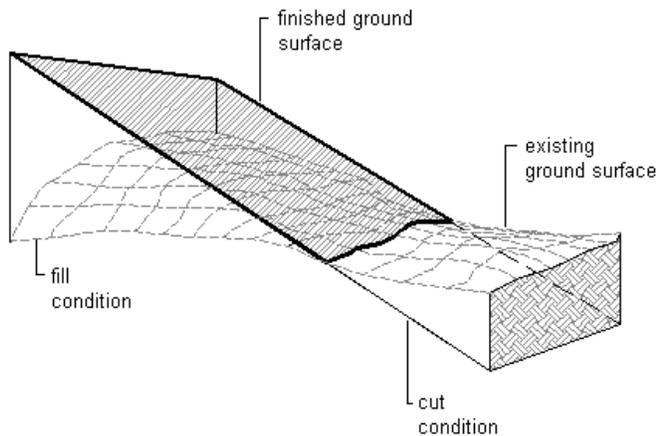


■ **Net**. The difference between the cut and the fill. For example, if a volume is 200 m3 of cut and 100 m3 of fill, the net is 100 m3 <cut>.

■ **Net Graph**. A graphical percentage representation of the whole volume. A fill net is displayed as a green bar indicating that material needs to be added to the project site. A cut net is displayed as a red bar, indicating that material must be removed.

## Quick Reference

Ribbon

Click Analyze tab ➤ Volumes And Materials panel ➤ Volumes drop-down ➤ Volumes 

Menu

Surfaces menu ➤ Utilities ➤ Volumes

Command Line

ReportSurfaceVolume

Dialog Box

Composite Volumes (page 2364)

# Calculating Bounded Volumes

Use the Bounded Volume utility to calculate the volume of an area as defined by a polyline, or polygon, or parcel.

You can quickly calculate and display the net volume, cut, and fill for a bounded area on a volume surface.

For volume surfaces, the volume of the bounded area is calculated based on the difference of elevations that were originally used for volume calculation. For terrain surfaces, the volume is calculated from the zero elevation to the elevations in the bounded area.

**See also:**

■ Creating a TIN Volume Surface (page 604)

■ Creating a Grid Volume Surface (page 606)

**To calculate bound volumes**

1  Open, create, or import the surface for which you want to measure the bounded volume. See Creating Surfaces (page 601).

2  Click Analyze tab ➤ Volumes And Materials panel ➤ Volumes drop-down ➤ Bounded Volumes 🔳 .

3  You are prompted to select the polyline, or polygon, or parcel that defines the bounded areas on the surface.

   The net volume, cut, and fill values are displayed at the command line.

   **TIP** Press F2 to open the AutoCAD Text Window where the results are also displayed.

## Quick Reference

Ribbon

   Click Analyze tab ➤ Volumes And Materials panel ➤ Volumes drop-down ➤ Bounded Volumes 🔳

Menu

   Surfaces menu ➤ Utilities ➤ Bounded Volumes

Command Line

   ReportSurfBoundedVolume

# Surface Styles and Visualization

Surface styles control the surface appearance in both 2D and 3D views.

Access and manage the surface styles in the Settings tab in Toolspace, where you can create, edit, copy, and delete them. For more information, see The Object Style Collection (Settings Tree) (page 94).

You can assign a surface style to a surface when you create or import a surface. You can also change the surface style using the surface list view.

The surface styles contain parameters and display settings for the creation of surface data objects, including:

■ **Borders**. Includes interior and exterior border and datum display.

■ **Contours**. Includes minor, major, depression, and user-defined contour lines display.

- **Grid**. Includes primary and secondary grid display.

- **Points**. All surface points for the TIN or grid surfaces.

- **Triangles**. TIN face information.

- **Analysis**. Includes directions, elevations, slopes, and slope arrows.

- **Watersheds**. Watersheds analysis display.

For general information about AutoCAD Civil styles, see Object Styles (page 53).

# Creating a Surface Style

Use the Settings tree to create a surface style.

A surface style defines how a surface is displayed in the drawing. You can apply the surface style to either a TIN surface, a grid surface, or a volume surface.

You can create and manage surface styles on the Settings tab in Toolspace.

**To create a surface style**

1 In Toolspace, on the Settings tab, right-click the Surface Styles collection and click New.

2 In the Surface Style dialog box, click the Information (page 2334) tab and enter a name and description for the surface style.

3 To define the display of surface borders, click the Borders (page 2335) tab and specify the settings.

4 To define the display of surface contours, click the Contours (page 2336) tab and specify the settings.

5 To define the display of surface grid, click the Grid (page 2338) tab and specify the settings.

6 To define the display of surface points, click the Points (page 2339) tab and specify the settings.

7 To define the display of surface triangles, click the Triangles (page 2341) tab and specify the settings.

8 To define the display of watersheds, click the Watersheds (page 2341) tab and specify the settings.

9 To define the display the analysis, click the Analysis (page 2343) tab and specify the settings.

10 To define the display of the various surface components, click the Display (page 2345) tab.

11 To view summary information about the style, click the Summary (page 2345) tab.

12 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Surface ➤ right-click Surface Styles ➤ New

Object Shortcut Menu

Edit Surface Style

Dialog Box

Surface Style (page 2334)

# Creating a Surface Style Based on an Existing Surface Style

Use the Settings tree to create a surface style based on an existing surface style.

You create a surface style based on an existing surface style by making a copy of the existing surface style and then editing it.

**To create a surface style based on an existing surface style**

1  In Toolspace, on the Settings tab, right-click the name of the surface style you want to copy and click Copy.

   A copy is created, and its properties are displayed in the Surface Style dialog box.

2  In the Surface Style dialog box, click the Information (page 2334) tab and enter a name and description for the new surface style

3  Use the Surface Style (page 2334) dialog box to change any of the surface style properties. For more information, see Creating a Surface Style (page 693).

## Quick Reference

Toolspace Shortcut Menu

   Settings tab: Surface ➤ Surface Styles ➤ right-click <style-name> ➤ Copy

Object Shortcut Menu

   Edit Surface Style

Dialog Box

   Surface Style (page 2334)

# Editing a Surface Style

Use the Settings tree to edit a surface style.

**To edit a surface style**

1  In Toolspace, on the Settings tab, right-click the name of the surface style that you want to edit and click Edit.

2  Use the Surface Style (page 2334) dialog box to change the properties of the surface style.

   For more information, see Creating a Surface Style (page 693).

## Quick Reference

Toolspace Shortcut Menu

   Settings tab: Surface ➤ Surface Styles ➤ right-click <style-name> ➤ Edit

Object Shortcut Menu

   Edit Surface Style

Dialog Box

   Surface Style (page 2334)

# Draping Images On Surfaces

Use the Drape Image command to overlay an image, such as an aerial photo, over a surface.

Before draping an image on a surface, insert the image into the drawing. The Drape Image command orients, aligns, and renders the inserted image so that the original flat image reshapes to replicate the relief of the surface terrain.

**To drape an image over a surface**

---

**TIP** To perform this operation, ensure that the surface display style has triangles visibility enabled. See Editing a Surface Style (page 694). Also, ensure that Realistic is a default type of materials set in the Material Editor in the Materials window (to view the settings, at the command line, enter MATERIALS).

---

1 Click Insert tab ➤ Reference panel ➤ Attach 📄 .

2 Navigate to the location where the image is saved, select the image, and click Open.

3 In the Attach Image dialog box, specify Insertion Point, Scale, and Rotation or select Specify on Screen to do the adjustments in the drawing.

4 Click OK.

5 Select a surface. On the Surface tab, Surface Tools panel, click Drape Image 🖾 .

6 In the Drape Image dialog box, select the image from the Image list or click 🖳 to select the image from the drawing.

7 From the Surface list, select the surface on which you want to drape the image or click 🖳 to select the surface from the drawing.

8 Under Render Material Name, enter the name for the new render material. This name will be associated with the new render material which is created from the image that you have imported.

---

**NOTE** By default, the name of the new render material is comprised of the surface name and the image file name.

---

9 Click OK.

The image is draped over the surface. This image becomes associated with the new render material, which is added to the materials palette.

If the image is smaller than the surface, the draping only occurs on the area covered by the image. If the area of the image exceeds the area of the surface, the image is cropped along the surface boundary.

---

**NOTE** If you view your drawing in 2D plan view, the image displays in its 2D planar view.

---

## Quick Reference

Ribbon

Select a surface. On theSurface tab, Surface Tools panel, click Drape Image 🖾

Menu

Surfaces menu ➤ Utilities ➤ Drape Image

Command Line

ImageAttach
DrapeImage

Dialog Box

Drape Image (page 2370)

# Surface Labels and Tables

Use surface label and table styles to control the appearance and behavior of surface-related labels and tables in a drawing.

The topics in this section describe what is unique to surface labels and tables. For a general overview of labels, see Labels and Tags (page 1463). For an overview of tables, see Tables (page 1555).

**Surface Labels**

Surfaces can be labeled automatically when created, using the specified label styles. After creating a surface, you can also add labels for the various components of the surface.

**Surface Tables**

You can use commands to insert surface legend tables into a drawing. Surface legend tables are used to display surface analysis information. When you insert a table into the drawing, the detailed surface information is inserted into the table.

# Surface Label Styles

Use the Settings tab of Toolspace to manage surface label styles. You specify which label styles to use when you create surfaces, when using the Add Labels dialog box to add slope and spot elevation labels.

For information about creating and using label styles, see Understanding Labels (page 1466).

You can create and use:

| This type of surface label style... | Labels... |
|---|---|
| Slope | A slope on a surface. The slope can be from a single point as it resides on a TIN face or surface grid cell, or between two points. |
| Spot elevation | A random point location with an elevation anywhere within a surface. |
| Watershed | The centroid of a watershed when the watershed component is displayed. |
| Contour | Contours at major, minor, and user-defined increments. |

## Slope Label Styles

A slope label style is used for labeling a slope on a surface.
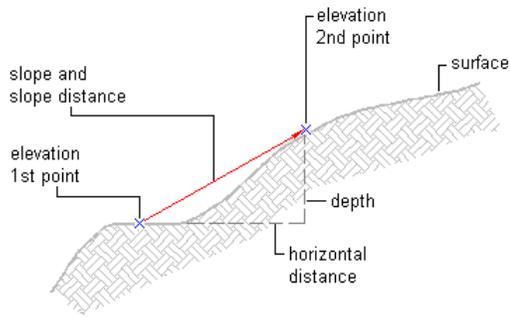
The slope can be labeled as a grade or slope, and is formatted using the slope label style. The direction of the slope is represented by a direction arrow component defined in the slope label style.

A slope label style requires these elements:

- Font (set in the Label Style Composer)

- Text height (set in the Label Style Composer)

- Orientation (set in the Label Style Composer)

- Directional arrow (set in the Label Style Composer)

- Position above/below directional arrow (set in the Label Style Composer)

- Attributes (set on the Properties tab of the Text Component Editor - Contents dialog box)

- Slope/Grade value

- Formatting/Precision (set in the Label Style Composer)

- Percent

- Decimal

- Run/Rise

## Spot Elevation Label Styles

Spot elevation labels are required for labeling a random point location with an elevation anywhere within a surface.

The spot elevation label style is configured in the Label Style Composer. When a spot elevation is created, the command can specify an existing point style and label style at create-time, or a block can be specified in the composition of the spot elevation label style.
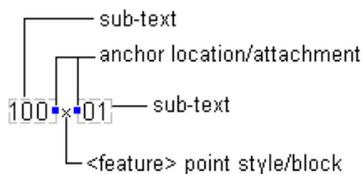


The following examples illustrate a label style that can label a spot elevation at the elevation decimal point. The decimal point of the spot elevation value is represented by a block, or a point style specified at creation time. The labeled elevation consists of two subtext components in the label style, each with different properties:

```
          ┌──  general
          │        anchor component: <feature>
          │        anchor point: middle left
          │        component name: elevation_left
          │    text
          │        attachment: middle right
          │        contents: <elevation[feet:0:normal:lhs]>
          │──  general
          │        anchor component: <feature>
          │        anchor point: middle right
          │        component name: elevation_right
─ 100·×·01 ─│    text
          │        attachment: middle left
          └──      contents: <elevation[feet:2:normal:rhs]>
```
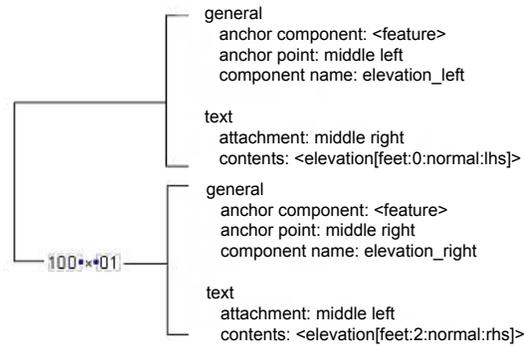
## Watershed Label Styles

Watershed label styles are used to draw a label at the centroid of a watershed when the watershed component is displayed, using the surface style.

Manage watershed label styles under the Surface collection in the Settings tree.

## Contour Label Styles

Use the Add Labels (page 1970) dialog box to add contour label lines to a surface.

Use the Add Labels dialog box to specify:

■ Labeling placement and spacing options

■ Major contour label style and increment properties

■ Minor contour label style and increment properties

■ User-defined contour label style

# Adding Surface Labels

You can label single slopes, spot elevations, and contours by using specified surface label types.

Watershed labels are created automatically. Their display is controlled by the surface style and cannot be added using this command.

## Adding Slope Labels

You can label individual slope points or label a slope between two points on the surface in the current drawing.

The slope can either be from a single point as it resides on a TIN face or surface grid cell, or between two points. The slope can be labeled as a grade or slope with a negative value representing a decreasing slope or grade. The direction of the slope is represented by a direction arrow component defined in the slope label style.

### Two-Point Slope Labels

You can create a slope label by selecting two points. The first point (XYZ) is the origin of the vector and the second point (XYZ) determines direction and slope.

**One-Point Slope Labels**

Create a slope label by selecting a single point. The normal of the surface triangle is determined and the selected point (XYZ) and the direction of the triangle normal are passed to the slope label style.

For more information about setting the slope label styles, see Slope Label Styles (page 696).

**To add slope labels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Slope 🖋 .

2   At the command line, enter either **One-point** or **Two-point** depending on the type of the slope label that you want to add and then press *Enter*.

3   Digitize the location of the first point and the second point (if applicable).

4   Optionally, repeat Step 3 to add more slope labels.

OR

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels 🖋 .

2   In the Add Labels dialog box, select Surface in the Feature list.

3   Select Slope from the Label Type list.

NOTE If you want to view or edit the settings for slope labels, click 🖾 to open the Edit Command Settings - AddSurfaceSlopeLabel dialog box.

4   Select the label style for the slope from the Slope Label Style list or use the standard style selection controls.

NOTE Ensure that the slope label style is appropriate for one-point or two-point slope labels. A slope label style that was composed for a start point and end point may not be appropriate for a one-point slope label.

5   Click Add.

6   At the command line, enter either **One-point** or **Two-point** depending on the type of the slope label that you want to add and then press *Enter*.

7   Digitize the location of the first point and the second point (if applicable).

8   Optionally, repeat Step 7 to add more slope labels.


## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Slope 🖋

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddSurfaceLabels
AddLabels

## Adding Spot Elevation Labels

Spot elevation labels are required for labeling a random point location with an elevation anywhere within a surface.

For more information about setting the spot elevation label styles, see Spot Elevation Label Styles (page 697).

**To add surface spot elevation labels**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Spot Elevation 🏷️ .

2 Select the point to label.

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels 🏷️ .

2 In the Add Labels dialog box, select Surface in the Feature list.

3 From the Label Type list, select Spot Elevation.

---

**NOTE** If you want to view or edit the settings for spot elevation labels, click 🗹 to display the Edit Command Settings - AddSurfaceSpotElevLabel dialog box.

---

4 Select the label style for the spot elevation from the Spot Elevation Label Style list or use the standard style selection controls.

5 Select the marker style for the spot elevation from the Marker Style list or use the standard style selection controls.

6 Click Add. You are prompted at the command line to select a point in the drawing area.

7 In the drawing, click the location of the point.

8 Optionally, repeat Step 7 to add more spot elevation labels.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Spot Elevation 🏷️

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddSurfaceSpotElevLabel
AddSurfaceLabels

## Adding Spot Elevation Labels On Grid

You can create a grid of surface spot elevation labels in the current drawing.

The following illustration shows a grid of surface spot elevation labels.
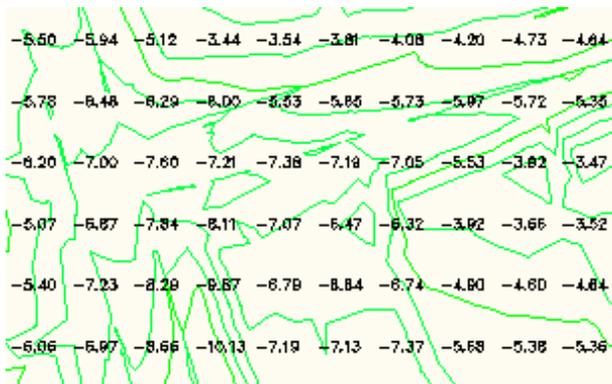
For more information about setting the spot elevation label styles, see Spot Elevation Label Styles (page 697).

**To add surface spot elevations on grid labels**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Spot Elevations On Grid 
    .

2  Follow command line instructions to add a grid spot elevation label.

OR

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels  .

2  In the Add Labels dialog box, select Surface in the Feature list.

3  From the Label Type list, select Spot Elevations on Grid.

---

**NOTE** If you want to view or edit the settings for spot elevation labels, click  to display the Edit Command Settings - AddSpotElevLabelsOnGrid dialog box.

---

4  Select the label style for the spot elevation from the Spot Elevation Label Style list or use the standard style selection controls.

5  Select the marker style for the spot elevation from the Marker Style list or use the standard style selection controls.

6  Click Add. You are prompted at the command line to select a point in the drawing area.

7  Follow command line instructions to add a grid spot elevation label.

8  In the drawing, click the location of the point.

9  Optionally, repeat Step 7 to add more spot elevation labels.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Spot Elevations On Grid 

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddSpotElevLabelsOnGrid
AddSurfaceLabels

## Adding Watershed Labels

Watershed labels can be considered an attribute of a watershed and not a separate component since they require no placement. Watershed labels are drawn at the watershed centroid.

For more information about setting the watershed label styles, see Watershed Label Styles (page 698).

## Adding Single Contour Labels

Use specified contour label styles to add single contour labels to a surface.

You can create contour labels by specifying line segments that cross contour lines of a surface. Add Single, Multiple, or Multiple at Interval contour labels. Modify the properties of contour label lines to change the label style for major, minor, and user-defined contour labels.

Use a line or polyline of an existing object as a label line or draw label lines on the fly for single or multiple contours.
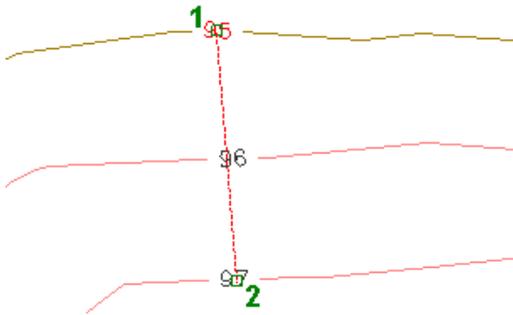


**NOTE** The line that is used to place the contour labels is a custom object called the Surface Contour Label Group object. The default layer for this object is set on the Object Layers tab of the Drawing Settings dialog box.

**TIP** Freezing the default layer of the contour label line freezes the contour labels. To control the visibility of the contour label line, change its display settings in the contour label line properties.

To make a contour label mask out the contour use the Background Mask property in the contour label style. Edit the contour label style using the Label Style Composer, and on the Layout tab under Border, set the value for Background Mask to True.

You can also use the Masking property in the AutoCAD Properties palette to mask contour lines. For more information, see Background Mask (page 1496).

**To add single surface contour labels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Single 🏷 .

2   Select the contour to label.

OR

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels 🏷 .

**2** In the Add Labels dialog box, from the Label Type list, select Contour - Single to add single contour labels by picking a single point on a surface contour line.

---

**NOTE** If you want to view or edit the settings for surface contour labels, click ⊞✓ to display the Edit Command Settings - AddContourLabelingSingle dialog box. Set the Display Contour Label Line to false to display contour label lines only when its label is selected.

---

**3** Depending on the type of surface contour lines (major, minor, or user-defined) that the contour label line crosses, specify the label styles for the corresponding contour lines:

- From the Major Contour Label Style list, select the style for major contour lines.

- From the Minor Contour Label Style list, select the style for minor contour lines.

- From the User Contour Label Style list, select the style for user-defined contour lines.

**4** Click Add.

**5** Click on the contour line where you want to place a label. You can add as many labels as you want. Press *Enter* when finished.

---

**6** **NOTE** You can set the default interval value. In the Add Labels dialog box, ⊞✓ to display the Edit Command Settings - AddContourLabelingSingle dialog box and set the Interval Along Contour property.

---

All the surface contours that were intersected by the contour label line are labeled at the specified intervals along their entire length.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Multiple At Interval ◔⁺

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddSurfaceContourLabelingGroup
AddSurfaceLabels

## Adding Multiple Contour Labels

You can use specified contour label styles to add multiple contour labels to a surface.

You can create contour labels by specifying line segments that cross contour lines of a specified surface. You can add multiple contour label types. Modify the properties of contour label lines to change the label style for major, minor, and user-defined contour labels.

Use a line or polyline of an existing object as a label line or draw a label line (1) that crosses multiple contours.
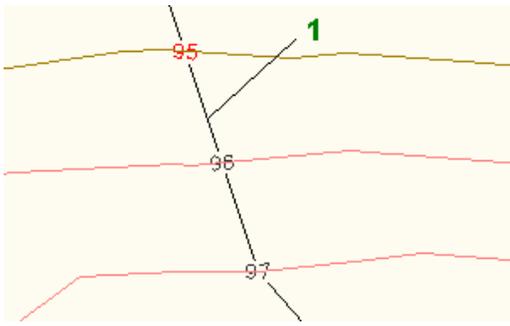
To make a contour label mask out the contour use the Background Mask property in the contour label style. Edit the contour label style using the Label Style Composer, and on the Layout tab under Border, set the value for Background Mask to True.

You can also use the Masking property in the AutoCAD Properties palette to mask contour lines. For more information, see Background Mask (page 1496).

**To add multiple surface contour labels**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Multiple ⬁.

2  Select the contours to label.

OR

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels ⬁.

2  From the Label Type list, select Contour - Multiple to add multiple contour labels either along the contour label line or polyline by picking segments to create Surface Contour Label Group objects from the line or polyline objects.

---

NOTE If you want to view or edit the settings for surface contour labels, click ⬚ to display the Edit Command Settings - AddContourLabeling dialog box. Set the Display Contour Label Line to false to display contour label lines only when its label is selected.

---

3  Depending on the type of surface contour lines (major, minor, or user-defined) that the contour label line crosses, specify the label styles for the corresponding contour lines:

   ■ From the Major Contour Label Style list, select the style for major contour lines.

   ■ From the Minor Contour Label Style list, select the style for minor contour lines.

   ■ From the User Contour Label Style list, select the style for user-defined contour lines.

4  Click Add.

5  Click on the contour line to specify the start point. Drag the contour label line across multiple contour lines until you reach the desired end point. Alternatively, after you specify the start point, you can click at individual points on the contour lines to create a contour label polyline. Press *Enter*.

---

6  NOTE You can set the default interval value. In the Add Labels dialog box, ⬚ to display the Edit Command Settings - AddContourLabeling dialog box and set the Interval Along Contour property.

---

All the surface contours that were intersected by the contour label line are labeled at the specified intervals along their entire length.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Multiple At Interval

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddContourLabeling
AddLabels

## Adding Multiple Contour Labels at an Interval

You can use specified contour label styles to add multiple contour labels to a surface at a specified interval.

Create contour labels by drawing a contour label line (1) that crosses surfaces contours and specify the interval (2) between the labels along those contours.
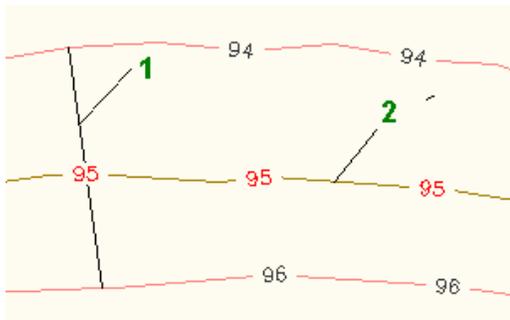


Modify the properties of contour label lines to change the label style for major, minor, and user-defined contour labels.

**To add multiple surface contour labels at an interval**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Multiple At Interval .

2 Click on the contour line to specify the start point. Follow the prompt in the command line to specify the next point on a contour label line and the interval along the contour line. Press *Enter*

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Add Surface Labels .

2 From the Label Type list, select Contour - Multiple at Interval to add multiple contour labels at a set interval along the entire length of the contour crossed by the label line segment.

NOTE If you want to view or edit the settings for surface contour labels, click to display the Edit Command Settings - AddContourLabelingGroup dialog box. Set the Display Contour Label Line to false to display contour label lines only when its label is selected.

**3** Depending on the type of surface contour lines (major, minor, or user-defined) that the contour label line crosses, specify the label styles for the corresponding contour lines:

   ■ From the Major Contour Label Style list, select the style for major contour lines.

   ■ From the Minor Contour Label Style list, select the style for minor contour lines.

   ■ From the User Contour Label Style list, select the style for user-defined contour lines.

**4** Click Add.

**5** Click on the contour line to specify the start point. Follow the prompt in the command line to specify the next point on a contour label line and the interval along the contour line. Press *Enter*.

---

**NOTE** You can set the default interval value. In the Add Labels dialog box, click ⊞ to display the Edit Command Settings - AddContourLabelingGroup dialog box and set the Interval Along Contour property.

---

All the surface contours that were intersected by the contour label line are labeled at the specified intervals along their entire length.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Surface ➤ Contour - Multiple At Interval ⬙

Menu

Surfaces menu ➤ Add Surface Labels ➤ Add Surface Labels

Command Line

AddContourLabelingGroup
AddLabels

# Modifying Surface Labels

Modify the display and location of surface and contour labels.

For information about modifying labels, see Inserting and Managing Labels in Drawings (page 1522). For more information about which properties of different label types can be modified, see Managing Layout Properties for Label Styles (page 1492).

# Surface Table Styles

Use the Toolspace Settings tab to manage surface table styles.

You can create and use the following types of surface table styles:

| This surface table style... | Places the following surface information in the table... |
|---|---|
| Direction | Aspect or direction ranges, colors, and area |
| Elevation | Elevation ranges, colors, area, and volume |
| Slope | Slope ranges, colors, and face areas |

| This surface table style... | Places the following surface information in the table... |
| --- | --- |
| Slope Arrow | Slope arrow ranges, colors, and face areas |
| Contour | Contour ranges, colors, areas, and volumes |
| Watershed | Number, type, description, color, hatching, and drainage area for the watersheds |
| User-Defined Contour | User-defined contour ranges, colors, areas, and volumes |

# Adding Surface Legend Tables

Use surface legend tables to organize and consolidate information about the surfaces in your drawing.

You can create surface legend tables for all types of surface analysis, including directions, elevations, slopes, slope arrows, contours, and watersheds. For more information about surface analysis, see Analyzing Surfaces (page 708).

You create and edit tables for most objects using the same common procedures and standard dialog boxes. For information about modifying tables, see Modifying Tables (page 1567).

You can control the display of the analysis in the drawing by adjusting the surface style Display settings.

Before creating a surface legend table, generate the surface analysis from the Analysis tab on the Surface Properties dialog box.

**To create a surface legend table**

1  Generate the surface analysis that you are going to display in the legend table. See Analyzing Surfaces (page 708).

2  To edit the styles for the tables, on the Settings tab, expand the Surface collection, expand the Table Styles collection, and select the collection for the type of legend that you want to add. In the Settings list view, right-click the style and click Edit.

3  To add the legend table, click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Add Surface Legend Table ⊞ .

4  You are prompted to select the surface from the drawing area.

5  You are prompted to enter the table type. Enter one of **Directions**, **Elevations**, **Slopes**, **SlopeArrows**, **Contours**, **UserContours**, or **Watersheds**.

6  You are prompted to specify if you want the table to update automatically if the analysis information changes. Enter **Dynamic** to enable automatic updating.

7  You are prompted to select the upper left-hand corner for the table in the drawing area.
   The legend table is displayed.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Add Surface Legend Table ⊞

Menu

Surfaces menu ➤ Add Legend Table

Command Line

AddSurfaceLegendTable

# Analyzing Surfaces

You can perform many types of surface-related analysis, including contour, directions, elevations, slopes, slope-arrows, watersheds, and water drop path.

Surface analysis types include:

- **Directions**. Used for aspect analysis. Renders surface triangles differently according to the direction they face.

- **Elevations**. Used for elevation banding analysis. Renders surface triangles differently according to their elevation range.

- **Slopes**. Renders surface triangles differently according to the slope range they fall within.

- **Slope Arrows**. Used for slope direction analysis. Places a slope directional arrow at each triangle centroid. The arrow color is based on the color assigned to a slope range, similar to slope analysis.

- **Contours**. Renders contour lines differently in according to their elevation range.

- **User-Defined Contours**. Renders user-defined contour lines differently according to their elevation range.

- **Watersheds**. Renders watersheds differently according to their type.

- **Contour Problems**. Used to locate problems with contours that are drawn according to the surface style contour settings.

- **Water Drop**. Used to trace the path that water would take across a surface.

Use the Analysis tab (page 2343) in the Surface Style dialog box to control the display and styles for direction, elevation, slope, and slope arrow analysis. Contours and watersheds have corresponding tabs (Contours tab (page 2336) and Watersheds tab (page 2341)), where their style and display are controlled. Use the Analysis tab (page 2349) in the Surface Properties dialog box to create the actual analysis.

To check for contour problems using a separate utility, click Analyze ➤ Ground Data panel ➤ Contour Check .

To perform water drop analysis in a separate utility, click Analyze ➤ Ground Data ➤ Water Drop .

# Creating Surface Analysis

Use the Analysis tab in the Surface Style dialog box to set up the analysis display and the Analysis tab in the Surface Properties dialog box to create the analysis.

**To perform surface analysis**

1   In Toolspace, on the Settings tab, right-click the Surface Styles collection and click New.

2   In the Surface Style dialog box, click the Information (page 2334) tab and assign a name and description to the style.

3   Click the Analysis (page 2343) tab and modify the properties as required.

**4** For contour analysis, to modify the default analysis values, click the Contours (page 2336) tab and modify the Contour Ranges property group as well as the Major and Minor display settings.

**5** For watershed analysis, click the Watersheds (page 2341) tab and modify the properties as required.

**6** Click the Display (page 2345) tab and modify the display settings so that the analysis is displayed correctly.

**7** Click OK to save the style and close the Surface Style dialog box.

**8** In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface for which you want to create an analysis, and click Properties.

**9** In the Surface Properties dialog box, click the Analysis (page 2349) tab and select the analysis that you want from the Analysis Type list.

> **NOTE** The Analysis tab displays varying fields and options depending on the type of analysis that you select.

**10** Optionally, select a legend style. For information about creating a legend, see Adding Surface Legend Tables (page 707).

**11** Modify the range or watershed parameters as required and click ⬇ to generate the analysis.

**12** Optionally, modify the details of the analysis by editing the fields in the Details table. For more information, see the Analysis (page 2349) tab.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Properties (Analysis tab)

Dialog Box

Surface Properties: Analysis Tab (page 2349)

# Checking for Contour Problems

Use the Check for Contour Problems utility to identify problems with contours that are drawn according to the surface style contour settings.

> **NOTE** The Check for Contour Problems utility operates on TIN surfaces only.

The utility takes into account the base elevation and interval specified in the surface style. When the utility encounters an invalid condition, a warning is displayed in the Event Viewer.

The utility looks at every surface point that falls on a contour. For each point, the utility counts the number of neighboring points (that is, points to which it is connected by an edge or a point that lies on the opposite side of a triangle that it belongs to) that fall at the same elevation. If the number of same-elevation neighbors is one, the warning "Contour ends at [x,y,z]" is displayed in the Event Viewer. If the number of same-elevation neighbors is more than two, the warning "Multiple contours at [x,y,z]" is displayed in the Event Viewer. In the second case, if the point is on the border of the surface, the warning "Multiple contours at border point [x,y,z]" is displayed.
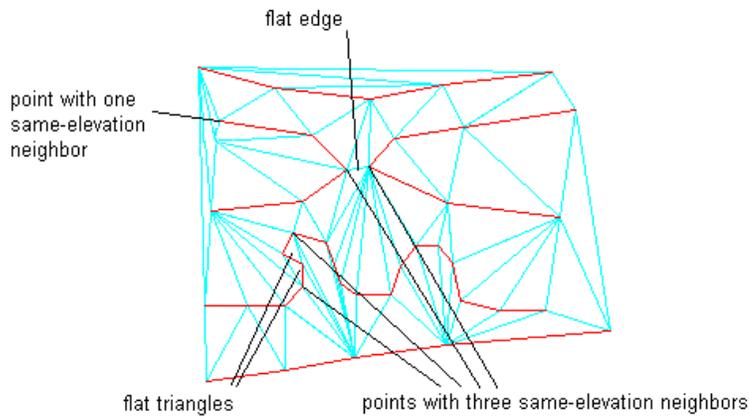
The "contour ends" condition can mean that a gap in the data contour was too wide for the triangulation process to connect the ends with an edge. You can fill this gap by creating and adding a breakline that connects the two ends.

The multiple contours condition, which occurs when a point has more than two neighboring points with the same elevation, generally happens in a flat area of the surface. When this condition is detected, you can use the Minimize Flat Areas operation to correct the problem. For more information, see Minimizing Flat Areas in a Surface (page 656). If the point is on the surface border, the flat area can simply be an unintended artifact of the triangulation process. You can remove these unintended triangles by using the Delete Line operation or by adding a hide boundary.

If no contour problems are found, a message is displayed in the Event Viewer stating that no problem were found for the surface style setting for contour base elevation and interval.

---

**NOTE** All contour problems occur when the elevation of a data contour point is the same as the elevation at which a contour is drawn. If no points in the surface are at contour elevations, there will be no problems with drawn contours. It is often possible and acceptable to adjust the contour base elevation value to create a condition where no points fall on surface contours, thus avoiding contour problems entirely.

---

**To check for contour problems**

1 Click Analyze tab ➤ Ground Data panel ➤ Contour Check  .

2 If more than one surface is available in the drawing, click the surface on which to perform the check or press Enter to select a surface in the Select A Surface dialog box.

3 Click OK.

Any detected contour problems are displayed in the Event Viewer.

## Quick Reference

Ribbon

Click Analyze tab ➤ Ground Data panel ➤ Contour Check 

Menu

Surfaces menu ➤ Utilities ➤ Check For Contour Problems

Command Line

CheckForContourProblems

# Drawing Water Drop Paths

Use the water drop utility to trace the path that water takes across a surface.

The utility draws a 2D or 3D polyline that represents a flow of water and also marks the start point of the path. If a channel splits, then new polylines are drawn to follow each water drop path.

For example, multiple water drop paths can be drawn from a contour line to illustrate the water flow at different points along the contour:
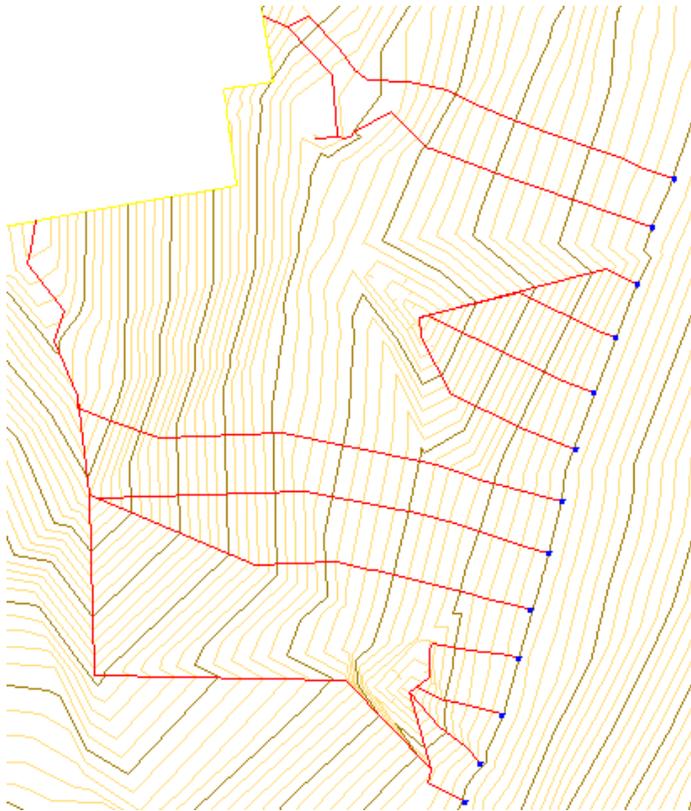


You can use either a 2D or 3D polyline to draw the flow line. The line type that you use depends on your analysis requirements. Following are usage examples for each line type:

- **2D polyline**. Delineate an area that represents a drainage area for water runoff analysis. You create a closed 2D polyline using AutoCAD editing commands. You can use the resulting closed 2D polyline to query the area and perform other analysis.

- **3D polyline**: You want to perform visualization and drape the lines over the surface. You can also create grading feature lines from the water drop lines for grading purposes.

**To perform water drop analysis**

1 Click Analyze tab ➤ Ground Data panel ➤ Water Drop 💧.

2 If more than one surface is available in the drawing, click the surface on which to perform the water drop analysis or press Enter to select a surface in the Select A Surface dialog box.

3 In the Water Drop (page 2367) dialog box, in the properties grid, click the Value column for the Path Layer property. Enter a name for the layer on which to draw the water drop polyline or click ⬚ to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

**4** To specify the type of polyline to use for the water drop path, click the Path Object Type field. Select the type.

**5** To specify whether to draw a marker at the start of the water drop path, click the Place Marker At Start Point field. Select either Yes or No.

**6** To set the style for the start point maker, click the Start Point Marker Style property. Click [···] in the Value column.

The Point Style dialog box is displayed.

**7** Click OK.

---

**NOTE** To be certain that any changes that you make to the water drop settings persist, ensure that the drawing setting Save Command Changes To Settings is set to Yes. For information, see Specifying Ambient Settings (page 68)

---

**8** Click a point on the surface for the start of the water drop path.

**9** Click another point to draw another water drop path or press Enter if finished.

### Quick Reference

Ribbon

Click Analyze tab ➤ Ground Data panel ➤ Water Drop 💧✳

Menu

Surfaces menu ➤ Utilities ➤ Water Drop

Command Line

CreateSurfaceWaterdrop

Dialog Box

Water Drop Dialog Box (page 2367)

## Displaying and Calculating Catchment Areas

Use the Catchment Area command to analyze water runoff and display the surface drainage area. The data obtained from running this command helps you perform H&H (hydrology and hydraulics) tasks, such as designing detention ponds, storm sewers, and other tasks aimed at the prevention of soil erosion.

The command defines the surface region with a depression low point (catchment point), delineates the region with a boundary, and calculates the area of the catchment region.

You can use a water drop path utility (page 711), to determine an accurate placement of catchment regions and catchment points.

---

**NOTE** The Catchment Area command performs the analysis only on the selected surface region and does not account for overflow situations occurring in the neighboring surface regions.

---

**To display a catchment region and calculate its area**

**1** Locate an area of interest in the drawing.

**2** Optionally, perform the following steps:

    **a** Click Analyze tab ➤ Ground Data panel ➤ Water Drop 💧✳ ..

    This command creates a polyline that represents the water drop path from a point on the selected region to the depression point in this region.

    **b** Select the water drop path polyline, right-click, and choose Properties.

    **c** In the Properties manager, under Geometry, navigate to the last vertex and copy and paste the X- and Y-coordinate values (comma-separated with no space) into a text file.

**3** Click Analyze tab ➤ Ground Data panel ➤ Water Drop 💧✳ ..

**4** In the Catchment Area dialog box, click the Value column for the Catchment Layer property and click ⊡ to select an existing layer, or create a layer in the Create Layer dialog box (page 1986).

> **BEST PRACTICE** It is useful to place the catchment region on a separate layer. Then you can control visual style elements, such as color and line types.

**5** Click the Value column for the Catchment Object Type, and select a line type.

> **TIP** If you select 2D Polyline, you can use the AutoCAD Area command on the catchment region. If you select 3D Polyline, you can use the elevation values of the polyline vertices for analysis.

**6** Click the Value column for the Catchment Marker property, and specify whether you want to mark the catchment point.

**7** Click ⊡ in the Value column of the Catchment Marker Style property, and then select a style in the Catchment Marker Style dialog box. Click OK.

**8** If more than one surface is available in the drawing, click the surface on which to perform the catchment region analysis, or press Enter to select a surface in the Select A Surface dialog box.

**9** Specify a catchment location. You can either paste the coordinates that you copied in Step 2 or click in the desired location.

    The catchment region boundary appears on the drawing, and the value of the region area displays at the command line.

**10** Repeat Step 11 for other coordinates or locations.

## Quick Reference

Ribbon

Click Analyze tab ➤ Ground Data panel ➤ Water Drop 💧✳

Menu

Surfaces menu ➤ Utilities ➤ Water Drop

Command Line

CatchmentArea

Dialog Box

Catchment Area (page 2372)

# Exporting Surface Data

You can export an AutoCAD Civil surface from a drawing to a DEM file compatible with either USGS or GeoTIFF formats, enabling you to use the surface in applications that support these formats.

## Exporting to DEM

You can export surface data to a DEM file and use the surface in other applications that support DEM data files.

Applications such as AutoCAD MAP 3D and AutoCAD Raster Design support both USGS and GeoTIFF standards. You can export all types of AutoCAD Civil surfaces, including terrain and volume surfaces.

**To export surface data to a DEM file**

1   Open a drawing with surfaces that you want to export to DEM.

2   Select a surface. Click Surface tab ➤ Surface Tools panel ➤ Export To DEM ⬆ ..

3   If more than one surface is available in the drawing, and you have not selected the surface in the previous step, in the drawing, click the surface which you want to export.

4   In the Export Surface to DEM (page 2368) dialog box, under Export, add information on the exported file, such as DEM file name, coordinate system, grid spacing, and elevation.

5   Specify if you'd like to use the default null elevation or specify a custom value and enter the value.

6   Click OK.

The surface data is imported to a DEM file in the location that you specified.

The exported DEM file inherits the coordinate zone assigned to the drawing. If the drawing does not have a coordinate zone assigned, you must specify one.

### Quick Reference

Ribbon

Select a surface. Click Surface tab ➤ Surface Tools panel ➤ Export To DEM ⬆

Menu

Surfaces menu ➤ Utilities ➤ Export To DEM

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Export to DEM

Command

SurfaceExportToDem

Dialog Box

Export Surface to DEM (page 2368)

## Exporting to LandXML

Export a TIN surface to a LandXML format file.

For more information about AutoCAD Civil LandXML import and export functionality, see Viewing and Editing LandXML Drawing Settings (page 1678).

**To export a surface to a LandXML format file**

1 In Toolspace, on the Prospector tab, expand the Surfaces collection, right-click the surface, and click Export LandXML.

The Export LandXML (page 1983) dialog box is displayed with the current surface selected for export.

2 To change the export selection set (for example, to add another surface), select or clear the check boxes in the data tree.

3 Click OK to export the surface to a LandXML format file.

4 In the Export LandXML dialog box, enter the name for the LandXML file.

The file is saved with the *.xml* extension.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Surfaces ➤ right-click <surface-name> ➤ Export LandXML

Dialog Box

Export LandXML (page 1983)

# Surfaces Command Reference

You can use commands to access surface functionality quickly.

The following table lists the surface-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AddContourLabeling | Adds multiple surface contour labels at points of intersection with contour label line (page 702) |
| AddContourLabelingGroup | Adds multiple surface contour labels at a set interval (page 702) |
| AddContourLabelingSingle | Adds a single surface contour label (page 702) |
| AddSpotElevLabelsOnGrid | Adds a spot elevation label on grid (page 700) |
| AddSurfaceBoundaries | Adds a boundary to an existing surface (page 615) |
| AddSurfaceBreaklines | Adds a breakline to an existing surface (page 618) |
| AddSurfaceContours | Adds contours to an existing surface (page 632) |
| AddSurfaceDEMFile | Adds a DEM file to an existing surface (page 637) |
| AddSurfaceDrawingObjects | Adds points from AutoCAD objects (page 640) |
| AddSurfaceGridPoint | Adds a grid point to an existing surface (page 653) |

| Command | Description |
|---|---|
| AddSurfaceLabels | Invokes a labeling tool used to add slope elevations or slope labels (page 698) |
| AddSurfaceLine | Adds a new TIN line to an existing surface (page 650) |
| AddSurfacePoint | Adds a point directly to the surface (page 653) |
| AddPointFile | Imports a point file into an existing surface (page 643) |
| AddSurfacePointGroup | Adds a point group to the surface definition (page 644) |
| AddSurfaceSlopeLabel | Adds a slope label to a surface (page 698) |
| AddSurfaceSpotElevLabel | Adds a spot elevation label to a surface (page 700) |
| CatchmentArea | Displays catchment regions and calculates the area of catchment regions (page 712) |
| CheckForContourProblems | Identifies Problems with Contours (page 709) |
| CreateSurface | Creates a TIN or Grid terrain and volume surface (page 601) |
| CreateSurfaceComposite | Calculates a composite volume between two surfaces (page 690) |
| CreateSurfaceGridFromDEM | Creates a grid surface from a DEM (Digital Elevation Model) file. (page 608) |
| CreateSurfaceFromTIN | Creates a TIN surface from a TIN data file (page 607) |
| CreateSurfaceReference | Creates a surface reference (page 609) |
| CreateSurfaceMask | Creates a mask on a surface (page 672) |
| CreateSurfaceStyle | Creates a new surface style (page 693) |
| CreateSurfaceWaterdrop | Creates a water drop path on the surface (page 711) |
| CreateSurfCntrLabelLine | Adds a contour label to a surface (page 702) |
| DeleteSurfaceGridPoint | Deletes a point from a grid surface (page 654) |
| DeleteSurfaceLine | Deletes a TIN line from a surface (page 651) |
| DeleteSurfacePoint | Deletes points from a TIN surface to remove inaccurate or unnecessary data (page 654) |
| DeleteSurfaceSlopeLabel | Deletes a slope label (page 706) |
| DrapeImage | Drapes an image onto a surface (page 695) |

| Command | Description |
| --- | --- |
| EditSurfacePaste | Pastes a surface into the current surface (page 662) |
| EditSurfacePoint | Modifies a surface point (page 654) |
| EditSurfaceProperties | Edits the properties of a surface (page 681) |
| EditSurfaceMask | Edits a surface mask (page 674) |
| EditSurfaceSettings | Displays the Edit Feature Settings - Surface dialog box (page 687) |
| EditSurfaceStyle | Edits the style of an existing surface (page 694) |
| EditSurfaceSwapEdge | Swaps the direction of two triangle faces (page 652) |
| MinimizeSurfaceFlatAreas | Minimizes the flat areas on a surface created from contours (page 656) |
| ModifySurfaceContourLabels | Modifies a contour label style (page 706) |
| ModifySurfaceSlopeLabel | Modifies a slope label style (page 706) |
| ModifySurfSpotElevLabel | Modifies a spot elevation label style (page 706) |
| MoveBlocksToAttribElev | Moves a block to an elevation value specified in one of its nested attribute values (page 641) |
| MoveBlocksToSurface | Moves a block object to a surface elevation (page 645) |
| MoveTextToElevation | Moves text to the elevation corresponding to its numeric value (page 642) |
| MoveSurfacePoint | Moves a surface point (page 655) |
| RaiseLowerSurface | Raises or lowers the entire surface (page 661) |
| RebuildSurface | Rebuilds a surface (page 685) |
| ReportSurfaceVolume | Displays composite volume information for two surfaces (page 690) |
| ReportSurfBoundedVolume | Calculates the volume of a bounded area (page 692) |
| SimplifySurface | Simplifies a surface by removing points (page 666) |
| SmoothSurface | Smooths the surface (page 662) |
| SurfaceModifyGridPoint | Modifies grid surface points (page 654) |
| SurfaceExportToDem | Exports surface data to a DEM file (page 714) |
| SurfaceExtractObjects | Extracts selected components from a surface (page 646) |

718

# Sites

# 20

In AutoCAD Civil, you can manage objects with a common topology in a collection called a site. The objects that share in the topology are parcels, alignments, gradings, and feature lines.

## Understanding Sites

Use sites to collect or group collections (parcels, alignments, gradings, and feature lines) by a topology that they have in common.

Examples of sites include soil maps, watersheds, or subdivisions. Sites can geographically overlay each other as is shown in the following illustration, while remaining independent. For example, a soils map and a subdivision with roads and lots can overlay each other, but remain independent of each other. Therefore, a parcel in a soils site would never be affected by an alignment in a subdivision site, even if their extents overlap.
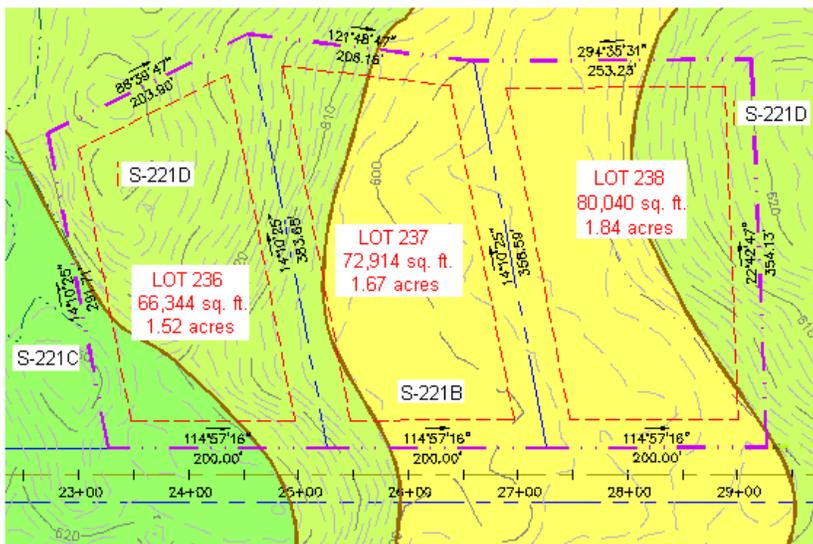


All objects in a site topology collection share a common topology and have relationships to each other.

**NOTE** Editing an object, or moving an object from one site to another, can cause changes to other objects that share the same topology.

When you create parcels and gradings, you specify a site for them. If you create these objects before explicitly creating a site, a default site named Site 1 is created for them.

When you create an alignment, you may either accept the default <none> site selection, place the alignment on an existing site, or create a new site. When an alignment is placed on a site, it will create parcels if it forms closed regions and subdivide existing parcels on the site if it crosses through them. Accepting the <none> site selection places the alignment in the top-level Alignments collection, which prevents it from forming or interacting with parcels or any other objects. See Alignment and Site Interaction (page 721) for more information.

**NOTE** An object, such as an alignment, parcel, or grading, can only exist in one site. All dependent objects, such as profiles and sections, exist in the same site as the parent object.

## Sites Collection

Use the Prospector tree to access the Sites collection, which can contain one or more site topology collections.

Each site topology collection contains collections for alignments, gradings and parcels:
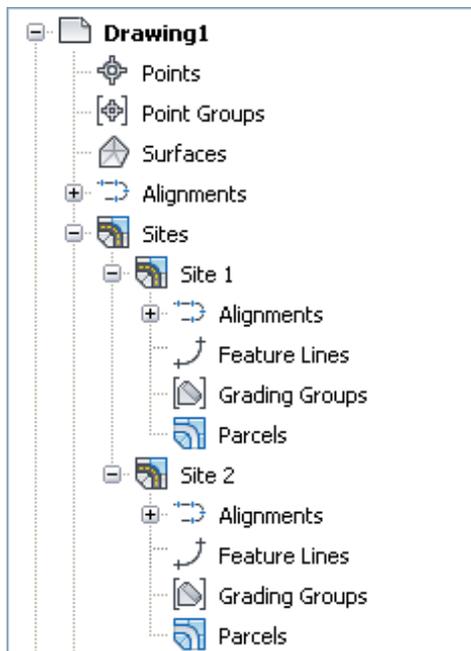


**NOTE** You can assign names to the site topology collections, but the Sites collection cannot be renamed.

Alignments may be assigned to either the Alignments collection within a topology collection or the top-level Alignments collection. When placed in the topology collection's Alignment collection, an alignment subdivides parcels it crosses over and creates new parcels if it forms closed regions. Alignments that are placed in the top-level Alignments collection do not interact with parcels or any other objects on any site. See Alignment and Site Interaction (page 721) for more information.

Expand the Sites collection node to view the sites in the collection. The site names with their respective area and perimeter values are displayed in the Prospector tab list view.

Access all site-related functionality by right-clicking the Sites collection and the named site topology collection (for example, Site 1 or Site 2) in the Prospector tree.

# Site Topology Collection

Use the named site topology collection (for example, Site 1) under Sites in the Prospector tree to edit or list the properties of a particular site, including its name, description, and 3D geometry.

Each named site topology collection contains collections of alignments, gradings, and parcels.

Right-click the named site topology collection to:

■ Edit and display the properties of the site, including 3D geometry and numbering conventions.

---

**NOTE** Parcel-related properties of a site are accessible from the site's Parcels collection. For more information, see Site Parcels (page 809).

---

■ Zoom or pan to the site.

■ Report on the site

■ Export the site to a LandXML format file.

■ Refresh the Prospector tree.

# Alignment and Site Interaction

When alignments are placed on a site, they will interact with other objects on the site.

Alignments interact with sites in two ways:

■ If an alignment exists on a site with parcels, the alignment will subdivide any parcels it crosses over.

■ If one or more alignments on a site form a closed region, a parcel will be created from the region.

Alignments may be assigned to either of two collections:

■ The top-level Alignments collection. When placed in this collection, an alignment will not interact with any other objects, including itself.

■ The Alignments collection within a topology collection, or site. When placed here, alignments will interact with other alignments and parcels on the site. If you want alignments to form parcels by closed regions but not subdivide existing parcels, assign alignments and parcels to separate sites.
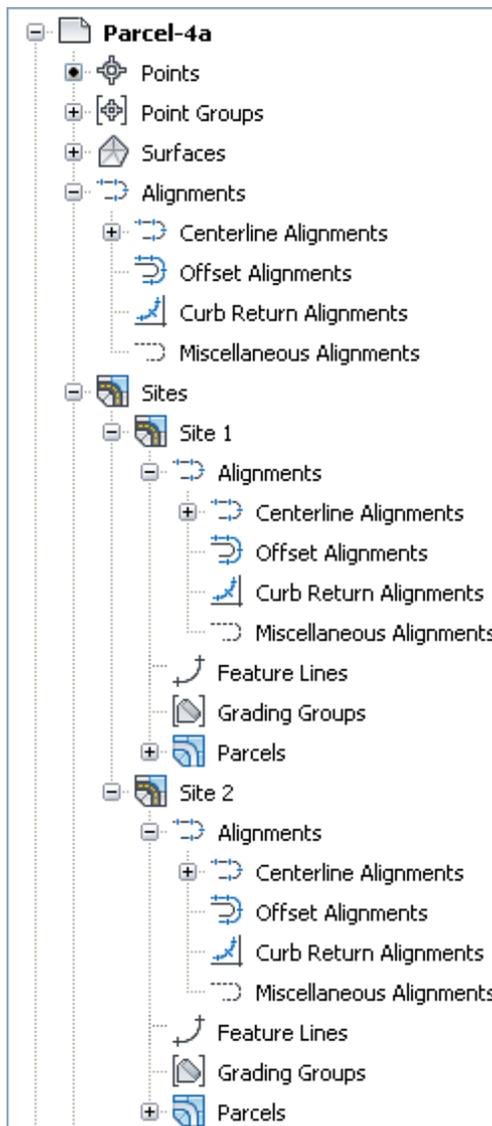
If you do not want an alignment to interact with other objects, accept the default <None> site selection when you create the alignment. Alignments can be moved or copied onto sites at a later time if interaction with other objects is desired.

## Importing from and Exporting to LandXML

You can import from and export to LandXML files.

When objects that are grouped under the Sites collection are exported to a LandXML file, the site name is exported with them.

When objects are imported from a LandXML file into the drawing, two site pickers are displayed:

■ **Alignments Site** allows you to specify the site on which the alignments will reside. The default selection is <None>, which places the alignments in the top-level Alignments collection and prevents them from interacting with parcels.

- Parcels Site allows you to specify the site on which the parcels will reside. If no sites exist in the drawing, the parcels are placed on the default site (Site 1). If the LandXML file has a site name for its features, then that site name is created in the drawing and the parcels are assigned to it.

For more information about importing to and exporting from a LandXML file, see LandXML Import and Export (page 1673).

# Managing Sites

After you create a site, you can edit its properties and add objects to it.

Before working with sites, you should review the following aspects of site functionality:

- Sites can be copied or moved. If there is a naming conflict at the destination site, the site name is automatically appended with the next sequential number.

- Objects can be copied or moved between sites. If there is a naming conflict at the destination site, the object name is automatically appended. For example, if the identical "Parcel A" object is copied three times into the same site, the name conflict is resolved as Parcel A (1), Parcel A (2), and Parcel A (3).

- Parcel linework, grading objects, and alignments can only belong to one unique site and do not interact with objects in a different site.

- Sites can only be deleted after all objects have been deleted from the site.

# Creating a Site

Use the Prospector tree to create a new site. You can create a site before creating the site topology (alignments, parcels, and grading).

**NOTE** When creating parcels and gradings, you must specify a site for them. You can create a new site when creating these objects. If you do not specify a site, a default site (Site 1) is created for them. You can rename the site using the Site Properties (page 724) dialog box.

**To create a site**

1 In Toolspace, on the Prospector tab, right-click the Sites  collection and click New.
   The Site Properties (page 2325) dialog box is displayed with the Information tab active.

2 Enter a name and description for the site in the corresponding fields.

3 Click the 3D Geometry (page 2325) tab and specify the 3D display settings for the site.

4 Click the Numbering  (page 2326)tab and specify the numbering properties for alignments and parcels.

5 Click OK.
   The site is added to the Sites collection in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: right-click Sites  ➤  New

Command Line

  CreateSite

Dialog Box

  Site Properties (page 2325)

# Editing Site Properties

A site has two types of properties: those that deal with the general site and those that are associated with the site but are primarily related to the parcel objects.

### General Site Properties

General site properties are applicable to all objects in a site. These properties include the site name, site description, 3D geometry display settings, and parcel and alignment numbering.

Access these properties by right-clicking the named site collection node (for example, Site 1) in the Prospector tree.
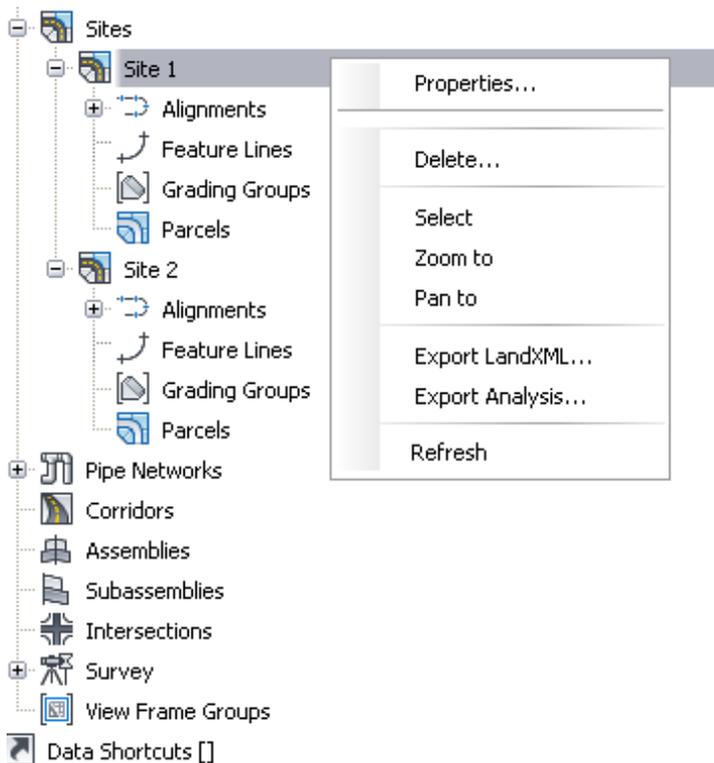


### Site Parcel Properties

Certain site properties are specific to the Parcels collection within a site. These properties include site parcel style, site area label style, parcel style display order, area, perimeter, and frontage.

Access these properties by right-clicking the Parcels collection under the named site collection.
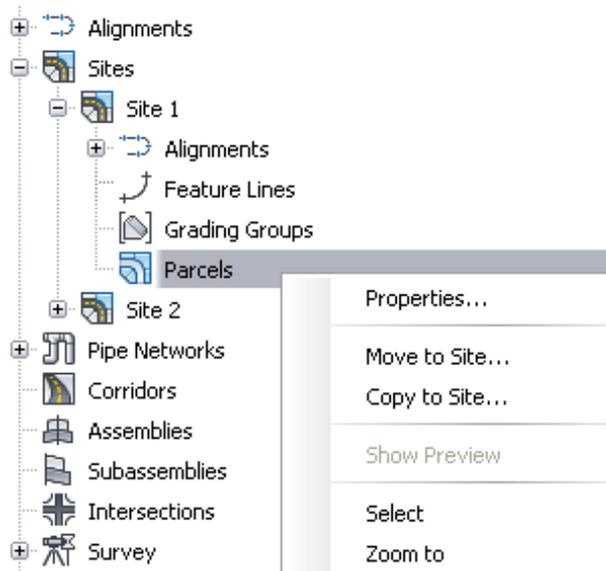
**NOTE** For information about editing Site Parcel properties, see Site Parcels (page 809).

**To edit site properties**

1   In Toolspace, on the Prospector tab, expand the Sites collection, right-click a site, and click Properties.

2   Optionally, on the Information (page 2325) tab, edit the name and description for the site.

3   Optionally, click the 3D Geometry (page 2325) tab and edit the 3D display settings for the site.

4   Optionally, click the Numbering (page 2326) tab and edit the numbering properties for the parcel components.

5   Click OK.

## Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Sites ➤ right-click <site-name> ➤ Properties

Command Line

   EditSite

Dialog Box

   Site Properties (page 2325)

# Specifying a Site for Topology Objects

When you create a parcel, alignment, or grading object, you are prompted to specify a site name.

Use the same site for all objects that share a common topology and have specific relationships to each other.

**NOTE** Editing an object could cause changes to other objects that share the same topology.

**To specify a site for topology objects**

1 Follow the specific procedure to create the object requiring a site (for example, parcel, alignment or grading object).

2 When the Create dialog box for the object is displayed, do one of the following:

- Click the Site list and select the site to which you want to add the object.
  When creating an alignment, select <None> to assign the alignment to the top-level Alignments collection. See Alignment and Site Interaction (page 721) for more information.

- Click the Down arrow next to the Site list and click Pick From Drawing.
  The following prompt is displayed:

  `Select object in site:`

  Select an object (parcel, alignment, or grading object) contained in the site to which you want to add the new object.

- Click the Down arrow next to the Site list and click Create New, then use the Site Properties (page 2325) dialog box to define the new site and click OK.

3 Perform the remaining object creation steps.

# Deleting a Site

Use the Prospector tree to delete a site. You must first delete all the objects that are contained in the site.

**To delete a site**

1 Delete any objects that are included in the site.

2 In Toolspace, on the Prospector tab, expand the Sites collection, right-click a site, and click Delete.
  The site is removed from the Prospector tree.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Sites ➤ right-click <site-name> ➤ Delete

# Moving and Copying

You can move or copy site objects from one site to another. The contents of an entire site, including alignments, grading groups, or parcels, can be moved or copied together or individually.

---

**NOTE** If there is a naming conflict, duplicate objects should be deleted from the destination site prior to moving or copying sites or objects within a site. If duplicates objects occur, the names are automatically resolved. For example, if the destination site contains an Alignment - (1), it is renamed to Alignment - (1) (1).

---

# Move Objects to a Site

Move site objects (alignments, gradings, and parcels) to a destination site, removing them from the source site.

Objects in the same site can interact with each other. To prevent this interaction, move one or more objects to another site.

Multiple sites can exist in the same geographic location.

Objects to move can be accessed from the Prospector tab for entire site collections or individual objects within the collection. A shortcut menu is also available from the drawing for valid objects within a site.

**To move objects to a site**

1 In Toolspace, on the Prospector tab, expand Sites.

2 Right-click a site, or collections or individual objects within the site, and click Move To Site.

3 In the Move To Site dialog box, under Destination Site, select an existing site from the list, or click to either pick a site in the drawing or create a new site.

When moving only alignments, select <None> to move the alignments to the top-level Alignments collection. See Alignment and Site Interaction (page 721) for more information.

4 Navigate in the data tree to view the selected objects and expand the collections. Select or clear the check boxes to filter the objects that you want to move.

5 Click OK. The selected objects are moved to the destination site. The original source site objects are removed.

## Quick Reference

Ribbon

Select the object. Click Modify panel ➤ ▼ ➤ Move To Site 

Toolspace Shortcut Menu

Prospector: Right-click object or object collection (Site, Alignment, Parcel, or Grading) ➤ Move To Site

Object Shortcut Menu

Drawing: Right-click object (Site, Alignment, Parcel, or Grading) object ➤ Move To Site

Command Line

MoveToSite

Dialog Box

Move To Site (page 2327)

# Copy Objects to a Site

Copy site objects (alignments, gradings, and parcels) to a destination site, while keeping the original objects at the source site.

Objects in the same site can interact with each other. Edits made to one object can affect other objects in the site.

Objects to copy can be accessed from the Prospector tab for entire site collections or individual objects within the collection. A shortcut menu is also available from the drawing for valid objects within a site.

**To copy objects to a site**

1 In Toolspace, on the Prospector tab, expand Sites.

2 Right-click a site, or collections or individual objects within the site, and click Copy To Site.

**3** In the Copy To Site dialog box, under Destination Site, select an existing site from the list, or click ![icon]
to either pick a site in the drawing or create a new site, which is then returned and displayed in the list.

When copying only alignments, select <None> to copy the alignments to the top-level Alignments collection. See Alignment and Site Interaction (page 721) for more information.

**4** Navigate the data tree to view the selected subcomponents and expand the collections. Select or clear the check boxes to filter the objects that you want to copy.

**5** Click OK. The selected objects are copied to the destination site. The original site objects remain in the source.

### Quick Reference

Ribbon

Select the object. Click Modify panel ➤ ▼ ➤ Copy To Site ![icon] .

Toolspace Shortcut Menu

Prospector: Right-click (Site, Alignment, Parcel, or Grading) ➤ Copy To Site

Object Shortcut Menu

Drawing: Right-click (Site, Alignment, Parcel, or Grading) object ➤ Copy To Site

Command Line

CopyToSite

Dialog Box

Copy To Site (page 2328)

# Sites Command Reference

The following table lists the site-related AutoCAD Civil commands and briefly describes their functionality. For more information about a command, follow the link in the Description column.

| Command | Description |
| --- | --- |
| CreateSite | Creates a site (page 723) |
| EditSite | Edits the properties of a site (page 724) |
| MoveToSite | Moves objects to another site (page 726) |
| CopyToSite | Copies objects to another site (page 727) |

# Grading

# 21

Use the grading tools and commands in AutoCAD Civil to design finished ground surfaces.

You create grading projections by applying grading criteria (page 2475) to a footprint. The grading objects are grouped into named collections to create your final surface designs and compute volumes.

You can also use the feature lines commands to create and edit 3D feature lines. Feature lines can be included directly into surfaces as breakline data, or you can use them as grading object footprints.

⊚ Grading Workflow

## Understanding Grading Objects

A grading object has its own properties and behavior, like other object types in AutoCAD Civil.

Before you start grading, you must configure settings and establish criteria such as the following:

- **Grading site**: Gradings are created in a site topology. If you do not want a grading to interact with other objects in a site, create a new site for your grading objects.

- **Grading group**: Grading objects in a grading group (page 2475) are consolidated to create one grading group surface so you can compute volumes. Before you create gradings, decide how you want to manage them with regard to surface creation and volume computations.

- **Grading footprints**: Grading objects are projected toward their target from a selected footprint. Footprints can be feature lines that you create specifically for this purpose, or you can export corridor feature lines or use parcel lot lines.

- **Grading targets**: Grading objects require a target. This can be a surface, a distance, or an elevation.

**Section view of surface as target**

surface as target

daylight

footprint
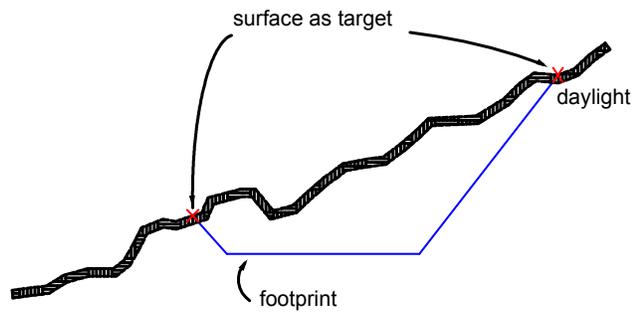
**Section view of elevation as target**

fixed elevation as target

footprint

X daylight

specified elevation
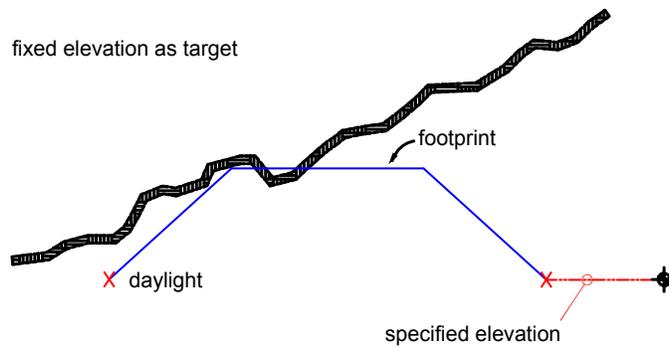
**Section view of relative elevation as target**

relative elevation as target

footprint

relative elevation

X daylight
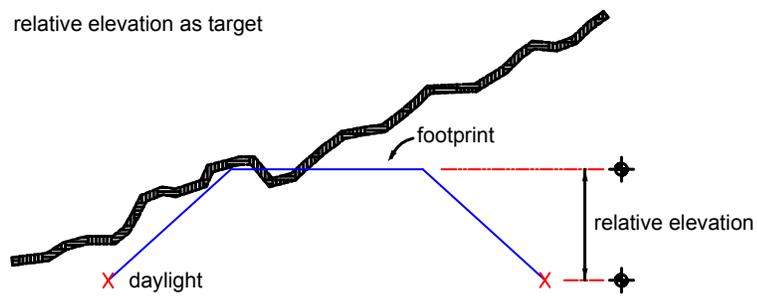
**Section view of distance as target**
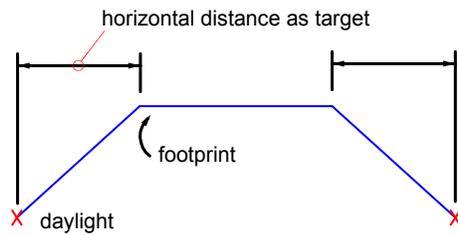


- **Grading criteria**: When you start grading, you specify grading criteria (page 2475). This is how many of the grading settings, such as grading target, are specified.

The list of grading objects defined in the drawing appears in Toolspace, within the Sites collection on the Prospector tab.

Grading styles (page 737) and Grading Criteria Sets appear in Toolspace on the **Settings** tab.

## Parts of a Grading Object

A grading object normally consists of the footprint, the daylight line, the projection lines, and the face.

The face is the area bounded by the lines that define the grading. It is marked with a center marker (page 2470).

**Plan view of grading object**
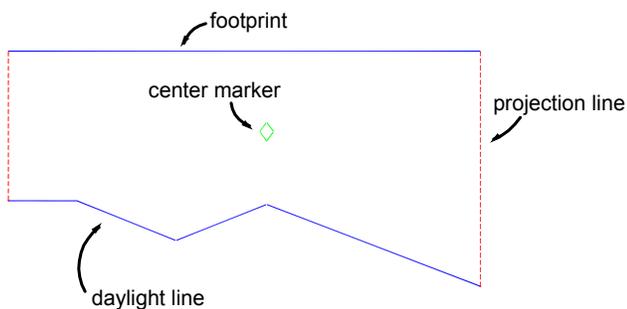


The footprint can be an open or closed figure. The footprint must be either a feature line (page 2473), a lot line, or the resulting daylight line (page 2472) of another grading.

A feature line is a line that represents some important feature in the drawing, such as a ridge line or the bottom of a swale. A feature line can be drawn with the Draw Feature Line command, or created from 2D and 3D polylines, lines, and arcs. You can also export feature lines from corridor models.

The target for the grading can be a surface, a distance, an elevation, or a relative elevation.

## Grading Regions

You define grading regions where a grading object requires different criteria along different parts of the footprint.
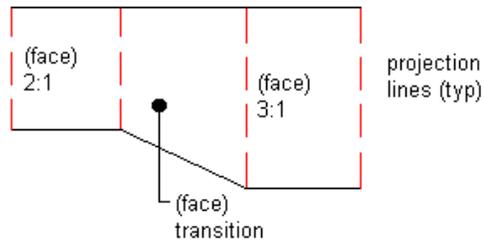
Where you need regions, you apply the criteria to only part of the footprint rather than the entire footprint.

For example, you might have a grading region with a slope of 3:1, and another region with a slope of 2:1, separated by a transition region, in which the grading merges from 3:1 to 2:1. Regions to which criteria have been applied are called control regions.

You are prompted to create a transition region if you specify the start point of a grading region to begin within an existing grading on the same footprint. You can also insert transition regions by using the Grading Creation Tools.

# Grading and Surfaces

Grading groups can be automatically turned into surfaces that dynamically update when you edit the grading. You can also create a detached surface from a grading group, which is no longer associated with the gradings and does not update when you edit the grading.

You can specify the Automatic Surface Creation setting when you create the grading group (page 741), or you can create the surface later by changing the Grading Group Properties (page 799). When you create an automatic (or dynamic) surface, the surface is displayed in the Surfaces collection in the Prospector tree. For dynamic surfaces, the surface properties show that it was created from a grading group.

### Editing Dynamic and Detached Surfaces

Dynamic surfaces created from grading groups are updated when you edit the grading object. If you turn on Rebuild Automatic in the Surface menu the surface updates as soon as you make the edit. You can also use the surface edit tools to refine the surface, such as adding spot elevations.

Detached surfaces created from grading groups are not updated when the grading object is edited.

You can edit both dynamic and detached surfaces like any other surface.

### Pasting Grading Surfaces

Both dynamic and detached surfaces can be pasted into other surfaces, except for the grading's target surface. By pasting a dynamic surface, any changes you make to the grading group will be reflected in the final surface.

---

**TIP** To paste a dynamic grading surface into another surface (the "destination surface"), you must select the Paste command from the destination surface node in the Prospector tree.

---

You cannot paste the grading surface into the surface that you are using as the target surface. This is disallowed because of the dynamic relationship between the gradings and the target surface. To accomplish this task you should create a copy of the target surface and paste the grading surface into the copy.

### Surface Breaklines and Tessellation Settings

When you create a surface from a grading group, the grading group faces are included as breaklines. The perimeter is included as a boundary.

The surface is built by adding extra breaklines between the footprint and daylight line for better surface definition. These extra breaklines are controlled by the tessellation settings, which you specify for each grading group. Tessellation parameters control the triangulation of surface (for both dynamic and detached surfaces) generated from the grading group.

Tessellation spacing represents the distance between the breaklines along the footprint segments. Tessellation angle represents the angular spacing of breaklines around corners.
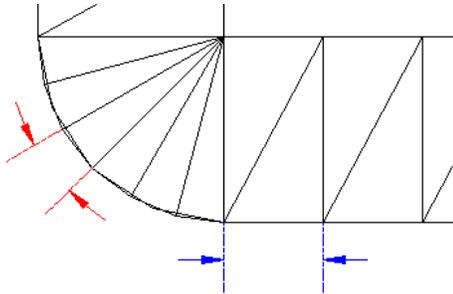
Finer values of spacing lead to a more accurate representation of the grading model, but increase the computation time. These values should be set according to the size of grading object in consideration.

### Calculating Volumes

To use the Grading Volume Tools, a dynamic surface must exist for the grading group and a Volume Base Surface must be specified in the Grading Group Properties. You can also use the surface volume commands to compute volumes between dynamic or detached surfaces and any other surface.

### Using Feature Lines as Surface Data

You can include feature lines as breakline data in a surface. Many of the feature line editing commands (page 751) also work on 3D polylines, parcel lines, and survey figures, giving you a wide variety of options for configuring the 3D data to include in finished ground surfaces.

If the Rebuild - Automatic setting for the surface is enabled, and you edit the feature line, parcel line, or polyline that is used as breakline data, the surface is rebuilt as soon as you make an edit.

# Feature Lines Collection (Prospector Tab)

Use the Feature Lines collection in the Prospector tree to manage feature line styles and layers. You can also view feature line information, such as length, elevation, and grade.

In Toolspace, on the Prospector tab, select the Feature Lines collection to display a list box of all the feature lines in the site. You can edit only the feature line style or layer values. Information such as length, elevation, and grade is read-only.
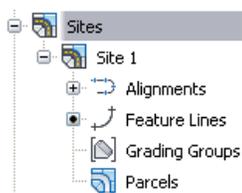
A black dot glyph indicates a feature line in the site. To edit the feature line style, expand the Site collection, right-click Feature Lines and select Properties. In the dialog box, set the feature line style priority on the Options tab (page 1915). If no feature lines have been created in the site, you can set the feature line style priority to determine the precedence for assigning elevations when feature line segments intersect. Then, when feature lines are created and points intersect, the elevations will be determined accordingly. You can

also set the default feature line style priority, using the command settings for Create Site. If feature lines exist in the site, you can also use the Properties command to view statistics about the feature lines.

Icons indicate the state of the feature line. ⬚ indicates a feature line with no dynamic links. ⬚ indicates a feature line with a dynamic alignment link. ⬚ indicates a feature line with a dynamic corridor link. For more information, see Creating Feature Lines (page 743).

These feature line icons display a small green square in the lower corner if the feature line is in use as a surface breakline ⬚ . You can find the name of the surfaces by listing or viewing the properties of individual feature lines. For more information, see Adding a Feature Line to a Surface as a Breakline (page 786).

You can edit a feature line Style or Layer value by clicking the cell to open a style or layer dialog box. To edit multiple values, use Ctrl+left-click to select multiple rows. Then right-click the column heading and click Edit to open a Select Style or Layer Selection dialog box. For more information about the AutoCAD Civil list view, see The Toolspace Item View (page 82).

Right-click the Feature Lines collection to do the following:

■ View properties for the site feature lines (page 801).

■ Apply feature line names (page 746).

■ Apply feature line styles (page 746). If the feature line does not have a style set, the context menu Edit Feature Line Style option is disabled. The style can be set in the Feature Line Properties dialog box.

■ Remove dynamic links to alignments (page 747). If there is no feature line dynamic link, the Remove Dynamic Links command is unavailable.

■ Raise or lower feature line elevations (page 764).

■ Add a feature line to a surface as a breakline (page 786).

■ Move all of the feature lines from this site to another.

■ Copy all of the feature lines in a site to another site.

■ Select all site feature lines (page 747).

■ Select all site grading groups (page 748).

■ Zoom to the site feature lines.

■ Pan to the site feature lines.

■ Refresh the view of the feature lines in the Prospector tree.

For more information, see The Toolspace Item View (page 82).

| For more information about ... | Follow this link ... |
| --- | --- |
| Feature Line Creation Methods | Creating Feature Lines (page 743) |
| Editing Feature Line Methods | Editing Feature Lines (page 751) |

## Grading Groups Collection (Prospector Tab)

As you create each grading group (page 2475) , they are displayed in the Grading Groups collection on the Prospector tab in Toolspace.

Right-click an individual grading group in the Grading Groups collection to do the following:

■ View properties for the grading group (page 799).

■ Delete the selected grading group.

■ Zoom to the extents of the grading group.

■ Pan to the site grading groups.

■ Select all gradings in the grading group (page 748).

■ Refresh the view of the grading group in the Prospector tree.

Expand the Grading Groups collection to view the names of the grading groups and to display a tabular list of the grading groups at the bottom of the Prospector tab. For more information, see The Toolspace Item View (page 82).

| For more information about ... | Follow this link ... |
| --- | --- |
| Grading Groups | Using Grading Groups (page 741) |
| Grading Creation Methods | Creating Grading (page 790) |
| Grading Editing Methods | Editing Grading (page 793) |

## Grading Collection (Settings Tab)

Use the Grading collection in the Settings tree to manage grading settings, grading styles, grading criteria sets, and grading command settings.

Right-click the Grading collection to do the following:

■ Edit the grading feature settings. (page 736)

■ Refresh the display of the Settings tree.

Expand the Grading collection to display and edit grading styles (including center markers and slope patterns), criteria sets, and command settings that are available for grading.

| For more information about ... | See ... |
| --- | --- |
| Grading Styles | Using Grading Styles (page 737) |
| Grading Criteria Sets | Using Grading Criteria (page 739) |
| Commands | Grading Command Reference (page 804) |

## Changing Grading Settings

Specify settings for all Grading commands with the Grading Feature Settings. Specify settings for specific Grading commands and override feature settings with the Grading Command Settings.

# Changing Grading Feature Settings

You can use the Grading Settings dialog box to change settings related to grading.

The settings you define in this dialog box override any ambient settings (page 1856) that have been established for the drawing as a whole. The General setting Save Command Changes To Settings is typically set to No. If you change it to Yes, then each time you run an affected command, the values you specify are saved as defaults for the next time you run the command.

**To change the grading feature settings**

1  In Toolspace, on the Settings tab, right-click Grading ➤ Edit Feature Settings.

2  In the Edit Settings (page 1875) dialog box, change the existing settings.

3  Click OK to save the changed settings in the drawing.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Grading collection ➤ Edit Feature Settings

Dialog Box

Grading Settings (page 1875)

# Changing Grading Command Settings

You can change the prompted values for a specific command, such as CreateGrading.

**TIP** The General feature setting Save Command Changes To Settings is typically set to No. If you change it to Yes, then each time you run an affected command, the values you specify are saved as defaults for the next time you run the command. This setting affects the grading criteria and styles in the Grading Creation Tools (page 1892) dialog box as well as the grading commands.

**To change default values for grading commands**

1  In Toolspace, on the Settings tab, expand Grading.

2  Expand the Commands collection and right-click a command name such as CreateGrading. Click Edit Command Settings.

3  In the Edit Settings (page 1875) dialog box, expand the command-specific property, such as Grading Creation, and edit the values.

4  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Expand Grading collection ➤ Commands ➤ right-click command name ➤ Edit Command Settings

Dialog Box

Grading Settings (page 1875)

# Using Grading Styles

You can use grading styles to control the display of each grading component.

Create styles to use for different types of gradings and for the various phases of your project. Create a style to use specifically in the design phase which shows lines, patterns and other grading subcomponents.

A style you use for plotting may show only the final grading geometry.

Grading styles are located on the Toolspace Settings tab. Slope pattern styles are located in the Multipurpose Styles collection because they are used for corridor objects as well as for gradings.

# Creating Grading Styles

Use the Grading Style dialog box to control the grading appearance.

Use the Slope Pattern Style (page 1901) dialog box to create different styles of slope patterns, which are referenced by grading styles.

You can create and save an unlimited number of grading styles. After you save the styles, they are located in the Grading Styles collection on the Toolspace Settings tab.

**To create a new grading style**

1  In Toolspace, on the Settings tab, right-click Grading Styles ➤ New.

2  In the Grading Style dialog box, click the Information tab (page 1894) and enter a name and optional description for the grading style.

3  To define the size of the center marker for a grading, click the Center Marker tab (page 1895).

4  To define the slope pattern for a grading, click the Slope Pattern tab (page 1895).

5  To define the display properties for the grading style, click the Display tab (page 1801) and select the display properties.

6  To view summary information about the style, click the Summary tab (page 1803).

7  Click OK to save the new style.

**To create a new slope pattern style**

1  In Toolspace, on the Settings tab, expand Multipurpose Styles ➤ Slope Pattern Styles.

2  Select Standard or another existing style that is similar to what you want. Right-click and click Copy. The Slope Pattern Style dialog box is displayed.

3  On the Information tab (page 1901), enter a name and optional description for the new style.

4  On the Layout tab (page 1902), edit parameter values for each component of the slope pattern and add or delete components as required.

5  Click OK to save the new style.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Grading Styles collection ➤ New
Settings tab: Multipurpose Styles collection ➤ Slope Pattern Styles ➤ Right-click <Style> item ➤ Copy

Dialog Boxes

Grading Style Dialog Box (page 1894)
Slope Pattern Style Dialog Box (page 1901)

# Setting the Current Grading Style

Specify a grading style to apply to each grading you create.

If you want the Grading Creation Tools dialog box (page 1892) to continually retain the style you last entered, in Toolspace, on the Settings tab, right-click Grading, select Edit Feature Settings, and specify Yes for the General setting Save Command Changes To Settings.

**See also:**

■ Changing Grading Command Settings (page 736)

**To specify a grading style**

1  Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools .

2  In the Grading Creation Tools (page 1892), click  to view the current criteria and the style.

3  Select the grading style that you want to apply from the list of all the styles defined in the drawing. When grading to a surface or elevation, there can be two style types, one for cut slopes and one for fill slopes.

4  Create the grading.

The grading style is applied to the grading and to any grading you create until you change the style by making another selection from the Grading Creation Tools.

# Editing Grading Styles

Use the Settings tree to edit existing styles, copy existing styles to create new ones, or delete styles.

**To edit, copy, or delete a grading style**

1  In Toolspace, on the Settings tab, expand the Grading collection. Expand the Grading Styles collection.

2  Right-click a grading style. Click Edit, Copy, or Delete.

If you are copying a style, use the Information tab in the Grading Style (page 1894) dialog box to change the name of the style before saving the style. The style appears in the Grading Styles collection with the new name.

You cannot delete a style that is currently applied to an object. First assign another style to the object and then delete the style.

## Quick Reference

Ribbon

Select the grading. Click Grading tab ➤ Modify panel ➤ Grading Properties drop-down ➤  Edit Grading Style

Toolspace Shortcut Menu

Settings tab: Right-click Grading Style item ➤ Edit
Settings tab: Right-click Grading Style item ➤ Copy
Settings tab: Right-click Grading Style item ➤ Delete

Dialog Box

Grading Styles (page 1894)

# Using Grading Criteria

You can create grading criteria, or parameters, to define how grading is created from the footprint, and apply the criteria to other gradings.

There are many standard grading situations. For example, you may frequently need to create grading with a slope of 3:1 and a target of a relative elevation. By defining a set of grading criteria with these values and saving it, you can easily apply these same values to any grading that you create.

# Creating Grading Criteria

Grading criteria are settings that specify the grading method. The settings are consolidated in named criteria to eliminate repetitive prompting when you are creating gradings.

The grading method settings include the following:

■ **Target**. Choices include surface, elevation, relative elevation, or distance.

■ **Projection type**. Choices include cut/fill slope, cut slope, fill slope, and distance.

■ **Interior corner overlap resolution**. Choices include averaging the slopes and increasing or decreasing one of the slopes. For more information and illustrations of the options, see Criteria Tab (Grading Criteria Dialog Box) (page 1883).

■ **Associated values**. Depending on the target and projection type, you can specify default elevations, distances, and cut/fill slopes to use.

Organize and access the criteria in the Settings tree, using the Grading Criteria Sets folders.

When setting up new criteria, it is recommended you use a descriptive naming convention. For example, the standard AutoCAD Civil drawing templates include several different criteria with names like "Surface @ 2-1 Slope." By using a clear naming convention, you do not need to review the criteria properties to know how it was defined.

**About Criteria Locking**

Click the icon to lock or unlock a criteria value. This criteria locking affects the prompts you receive and the values you can enter when creating or editing grading.

Lock criteria values to prevent them from being changed when you create or edit gradings.

For example, consider the different lock conditions for a criteria that is defined to grade at a slope of 3:1 to a relative elevation of 1.50:

**Both values are unlocked**

■ When creating a grading, you are prompted for both values. The prompts default to 3:1 and 1.50, but can be changed.

■ Either value can be edited.

■ If the criteria definition is edited to change either value, it affects only the default prompt value for grading creation; existing grading objects do not change.

■ If the criteria definition is edited and one of the values is changed from unlocked to locked, that locked value is applied to all gradings that use the criteria.

**Slope value is locked, relative elevation is unlocked**

■ When creating grading, you are prompted only for the relative elevation value. Grading objects that use this criteria always have a 3:1 slope.

■ If the criteria definition is edited to change the locked slope value to 2:1, the change applies to all grading objects that use that criteria. If the criteria definition is edited to unlock the slope value, it does not change existing gradings, but allows you to edit the slope value for existing grading objects.

■ If the criteria definition is edited to change the unlocked relative elevation value, it affects only the default prompt value for new grading objects created with the criteria. If the criteria definition is edited to lock the relative elevation, all grading objects using that criteria are set to the locked relative elevation value.

**Both values are locked**

■ When creating grading, you do not receive prompts for the criteria values.

■ If the criteria definition is edited to change either value, the change applies to all grading objects that use the criteria.

■ If the criteria definition is changed to unlock either value, it does not change the grading objects but allows you to edit the value for existing grading objects that use the criteria.

**To create a new grading criteria set**

**1** In Toolspace, on the Settings tab, expand Grading.

**2** Right-click the Grading Criteria Sets collection and click New.

**3** In the Grading Criteria Set Properties (page 1888) dialog box, enter a name and optional description for the set and click OK.

**To create a new grading criteria**

**1** In Toolspace, on the Settings tab, expand Grading.

**2** Expand the Grading Criteria Sets collection. The defined sets are listed.

**3** Right-click a named grading criteria set. Click New.

**4** In the Grading Criteria dialog box, click the Information tab (page 1882) and enter a name and an optional description.

**5** Click the Criteria tab (page 1883), and edit the values to define the criteria you want.

**6** Lock the items that you do not want to change when you are creating grading objects. For example, if you are setting up a criteria to always create a 3:1 slope, lock the Slope value by clicking the lock icon.

**7** Click OK.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: Right-click Grading Criteria Sets collection ➤ New
Settings tab: Right-click a grading criteria set ➤ New

Dialog Box

Grading Criteria (page 1882)

# Setting Default Grading Criteria

Apply a set of grading criteria to each grading that you create.

**To set default grading criteria**

1  Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools ⬚ .

2  In the Grading Creation Tools (page 1892) dialog box, click ⬚ (Select A Criteria Set).

3  Select the grading criteria that you want to apply.

   Click ⬇ if you want to view the settings for the current criteria.

4  Create the grading.

   The grading criteria is applied to each grading you create until you change criteria by making another selection from the Grading Creation Tools.

# Using Grading Groups

You can specify which collection a grading object is part of.

Once you create a surface from a grading group, you can use the Grading Volume Tools (page 1899) to adjust the elevation of the grading group to balance cut and fill volumes.

# Creating a New Grading Group

Use the Prospector tab of Toolspace to create a new grading group.

Grading groups are used to organize gradings into named collections for surface creation and volume computations.

You can choose whether to automatically create a surface from the grading group, and you can identify a base surface for volume calculations.

You can also create new grading groups while using the Grading Creation Tools command.

**To create a new grading group**

1  In Toolspace, on the Prospector tab, expand Sites, then expand the site to which you want to add the new grading group.

2  Right-click Grading Groups ➤ Create Grading Group.

3  In the Create Grading Group (page 1881) dialog box, enter a name and an optional description for the grading group.

**4** Optionally, to create a surface for the grading group, select Automatic Surface Creation to activate additional options for the output surface and the base surface. It also creates a surface in the Surfaces collection in the Prospector tree.

> **NOTE** When you create a surface using the Automatic Surface Creation option, the Grading Group is listed on the Definition tab of that surface's Properties dialog box.

**5** Optionally, if you plan to do any calculations of cut and fill volumes for the grading group surface, select Volume Base Surface and select the base surface from the list.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Right-click Grading Groups collection ➤ Create Grading Group

Command Line

CreateGradingGroup

Dialog Box

Create Grading Group (page 1881)

## Changing the Grading Group of a Grading Object

A grading object can be moved from one grading group to another.

If you do not want certain grading objects to be included in the same grading group, you can move them to a different group. There must be at least two grading groups in the same site.

**To change the grading group of a grading object**

**1** Select a grading. Click Grading tab ➤ Modify panel ➤ Change Group  .

**2** Select one or more grading objects in the drawing.

**3** In the Select Grading Group (page 1900) dialog box, select or create a grading group for the grading object to be in.

**4** Click OK.

### Quick Reference

Ribbon

Select the grading. Grading tab ➤ Modify panel ➤ Change Group 

Menu

Grading menu ➤ Edit Grading ➤ Change Group

Command Line

ChangeGradingGroup

Dialog Box

Select Grading Group (page 1900)

# Creating Feature Lines

You can select a parcel lot line or feature line (page 2473) as the grading footprint.

A feature line is a special type of line that grading commands recognize and use as a footprint.

A feature line represents an object in the drawing from which you want to grade, such as a swale or a ridge line. You can draw feature lines, create them by converting existing objects, or export feature lines from corridors. Surfaces can use a feature line as a breakline.

### Curved Feature Lines

Unlike 3D polylines, feature lines support arcs without tessellation (page 2484). Tessellation is undesirable in a grading footprint because it results in many small grading faces joined by radial corners. To avoid tessellation, you can create a feature line from a 2D polyline with arcs, and then apply elevations using theGrading Elevation Editor. If you want to create a grading from a footprint that has tessellated curves, you can use the Fit Curve (page 777) command to convert the tessellation to true arcs.

### See also:

- Exporting Corridor Feature Lines (page 1369)

# Creating Feature Lines from Objects

You can create a feature line by converting existing 2D or 3D polylines, lines, or arcs. You can select objects in the current drawing or from an Xref.

Names can be assigned to feature lines. Any command that prompts to select a feature line includes an option to select by name from a list. The name is optional so that you can just name the significant feature lines in your drawing.

You can specify the CreateFeatureLines (page 736) command settings prior to creating feature lines.

Settings control which layer the lines are placed on and whether the original objects are deleted.

**To create a feature line from existing objects**

1 Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects ⬚ .

2 In the drawing, select one or more polylines, lines, or arcs. Then press Enter.

NOTE You can enter Xref to select an object from an Xref.

You can select the objects one at a time, or you can make a multiple selection by drawing a window.

3 In the Create Feature Lines (page 1907) dialog box, specify the feature line options.

The feature line retains the elevations of the objects from which it was made. To edit these elevations, see Editing the Elevations of a Feature Line (page 753).

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects

Menu

Grading menu ➤ Create Feature Lines From Objects

Command Line

CreateFeatureLines

# Assigning Elevations to a Feature Line During Creation

When creating a feature line you can manually enter an elevation or assign an elevation from a surface or from grading objects when creating a feature line from objects.

**To manually assign feature line elevations**

1   Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects .

2   In the Create Feature Lines  (page 1907) dialog box, select the Assign Elevations check box and press OK.

3   In the Assign Elevations (page 1908) dialog box, enter an elevation in the edit box select the Elevation option and press OK. If the object is a 3D polyline with varying elevations, the first point is set to the specified elevation and the rest of the points are raised/lowered by the same relative amount so that the elevation differences between the points remain.

**To assign feature line elevations from a surface**

1   Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects .

2   In the Create Feature Lines dialog box (page 1907), select the Assign Elevations check box and press OK.

3   In the Assign Elevations (page 1908) dialog box, select the From Surface option.

    **NOTE** If there are no surfaces in the drawing, the From Surface option is unavailable.

4   Select a surface from the list or click the arrow icon to select it in the drawing.

5   Select the Insert Intermediate Grade Break Points check box to insert intermediate grade breaks where the entity crosses surface TIN lines. Additional elevation points are created at these locations.

**To assign feature line elevations from a grading in the current site**

1   Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects .

2   In the Create Feature Lines dialog box (page 1907), select the Assign Elevations check box.

3   Select the From Gradings option. This option is similar to the From Surfaces option but acquires the elevations directly from any gradings that it overlaps in the site that the feature line is to be created in.

    **NOTE** If there are no gradings in the drawing, the From Gradings option is unavailable.

**4** Select the Insert Intermediate Grade Break Points check box to insert intermediate grade breaks where the entity crosses surface TIN lines. Elevation points are created at these locations.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects

Menu

Grading menu ➤ Create Feature Lines From Objects

Command Line

CreateFeatureLines

# Creating Feature Lines from Alignments

You can create a feature line from an alignment, and use this feature line to grade from the alignment geometry.

You can also create a link from the feature line to the alignment so that it updates dynamically if the alignment is edited. If the feature line is linked to the alignment, you cannot edit it directly.

However, it updates automatically when you make edits to the alignment or its profile. These edits also update any gradings attached to the feature line. If the feature line is not dynamically linked, it does not maintain its relationship with the alignment and you can edit it.

You can specify the CreateFeatureLineFromAlign (page 736) command settings prior to creating feature lines.

**To create a feature line from an alignment**

**1** Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects .

**2** In the drawing, select an alignment. Then press Enter.
If there are no sites in the drawing, a site will automatically be created.

**3** In the Create Feature Line From Alignment (page 1909) dialog box, specify the feature line options.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Lines From Objects

Menu

Grading menu ➤ Create Feature Lines From Objects

Command Line

CreateFeatureLineFromAlign

# Applying Feature Line Names

Apply optional names to feature lines to distinguish significant feature lines.

After creation, you can edit the names of individual feature lines using the Properties command, or using the grid control in Prospector.

This command is used to apply a name template to a selection set of feature lines.

You can specify the CreateFeatureLines (page 736) command settings prior to creating feature lines.

**To apply a feature line name**

1  In the drawing, select a feature line, right-click, and click Apply Feature Line Names.

2  In the Apply Feature Line Names (page 1915) dialog box, in the Name field, enter a name for the feature line.

   To name the feature line, either select its default name and enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

## Quick Reference

Ribbon

Select the feature line. Click Feature Line tab ➤ Modify panel ➤ Apply Feature Line Names

Menu

Prospector tab ➤ Feature Lines ➤ Apply Feature Line Names

Command Line

ApplyFeatureLineNames

# Applying Feature Line Styles

Apply a style to a selected set of feature lines.

You can specify the CreateFeatureLines (page 736) command settings prior to creating feature lines.

**To apply a feature line style**

1  In the drawing, select two or more feature lines, right-click, and click Apply Feature Line Styles.

2  In the Apply Feature Line Styles (page 1916) dialog box, specify the feature line style.

## Quick Reference

Ribbon

Select the feature line. Click Feature Line tab ➤ Modify panel ➤ Apply Feature Line Styles

Menu

Prospector tab ➤ Feature Lines ➤ Apply Feature Line Styles

Command Line

ApplyFeatureLineStyles

# Editing Feature Line Styles

Apply a style to a selected set of feature lines.

Controls the display of each feature line component.

### To edit a feature line style

1   In the drawing, select a feature line, right-click, and click Edit Feature Line Style.

2   In the Feature Line Style Dialog Box (page 1896), specify the feature line style components.

## Quick Reference

Ribbon

Select the feature line. Click Feature Line tab ➤ Modify panel ➤ Feature Line Properties drop-down ➤ Edit Feature Line Style ⊷⊡

Command Line

EditFeatureLineStyle

# Removing Dynamic Links

Remove the dynamic links from grading feature lines that have been created from alignments or corridors.

Removing a dynamic link disconnects the feature line from an alignment or corridor and enables feature line editing. You cannot edit feature lines with dynamic links. When a dynamic link is removed, it cannot be restored.

### To remove a dynamic link

1   In the drawing, select a feature line, right-click, and click Remove Dynamic Links.

2   In the Apply Feature Line Styles (page 1916) dialog box, specify the feature line style.

## Quick Reference

Menu

Prospector tab ➤ Feature Lines ➤ Remove Dynamic Links

Command Line

FeatureLineRemLinks

### See also:

■   Creating Feature Lines (page 743)

# Selecting All Feature Lines in a Site

Make a selection set of all feature lines in the current site and edit them all simultaneously. The same command also displays the Toolspace List View.

**To select all feature lines in a site**

**1**  In Toolspace, on the Prospector tab, expand the Site collection.

**2**  Right-click Feature Lines and click Select.

**3**  All features lines are selected.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab ➤ Feature Lines ➤ Select

## Selecting All Grading Groups in a Site

Make a selection set of all grading groups in the current site. The same command is useful for visualizing the gradings in the object viewer. This command also displays the Toolspace List View.

**To select all grading groups in a site**

**1**  In Toolspace, on the Prospector tab, expand the Site Collection.

**2**  Right-click Grading Groups and click Select.

**3**  All grading groups are selected.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab ➤ Grading Groups ➤ Select

## Drawing Feature Lines

Draws feature line segments.

Use this command to draw straight and curved feature line segments. This command has options to specify the elevations of the feature line, including an option to retrieve the elevations from a surface.
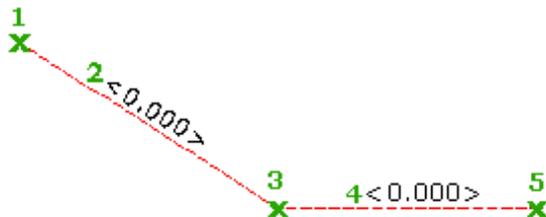


Arcs are created tangential to the previous segment except when the Second Point option is selected to create a three-point arc.

**To draw straight segments of feature lines**

1 Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line 🛠 .

2 In the Create Feature Lines  (page 1907) dialog box, specify a site, an optional name, and an optional style for the feature line. Select a layer and note that the option to use a selected entity layer is disabled. Click OK.

3 Select the start point for the line. You can use Osnaps or transparent commands to snap to existing geometry.

4 Specify the elevation for the start point by doing one of the following:

■ Enter an elevation at the command line.

■ Enter Surface (S) and select a surface from which to obtain the elevation.

> **NOTE** If there are no surfaces in the drawing, the Surface option is not available. If there is only one surface in the drawing, that surface is selected automatically.

The following prompt is displayed:

```
Specify the next point or [Arc]:
```

5 Select the next point.

The following prompt is displayed:

```
Specify grade or [SLope/Elevation/Difference/SUrface/Transition] <0.000>:
```

> **NOTE** See To draw curve segments of feature lines (page 750) for information about the Arc option.

6 If you wish to specify each elevation point as you create points, skip down to Step nine. To continue drawing points without establishing the elevation until the end:

■ Enter the Transition command.

■ Create a new point. When Transition is selected, as each subsequent point is created the elevation prompt will default to Transition.
```
Transition or [SLope/Elevation/Difference/SUrface] <Transition>
```

7 Press **Enter** to continue drawing another point(s) without specifying the elevation.

8 After creating points, select and enter the desired keyword to specify the elevation, grade, slope, or elevation difference. If you end the command while still in the Transition prompt, you will be prompted to specify an ending elevation.

9 Do one of the following to establish the elevation of the second point:

■ Enter a grade. The grade is applied to the segment between the first and second points.

■ Enter Slope (SL) and enter a slope. The slope is applied to the segment between the first and second points.

■ Enter Elevation (E) and enter an elevation. The elevation is applied to the second point and the grade of the segment between the points is calculated.

■ Enter Difference (D) and enter an elevational difference between the first and second points.

■ Enter Surface (SU) and specify a surface from which to obtain the elevation of the second point.

10 Specify the next point on the feature line by doing one of the following:

■ Select the next point in the drawing.

■ Enter Length (L) and enter a length. When you use this option, the direction of the segment is assumed to be the same as the end of the previous segment.

11 Specify the elevation of the point using the options described in Step 6.

**To draw curve segments of feature lines**

1 Complete Steps 1-4 in Drawing Feature Lines (page 748) to draw straight segments.

2 When you are prompted to select the next feature line point, enter Arc (A). The following prompt is displayed:

```
Specify arc end point or [Radius/SecondPnt/Line/Undo]:
```

**NOTE** If you have not drawn any feature line segments, you are first prompted to establish the arc direction. For more information, see To establish arc direction (page 750).

3 Do one of the following:
■ Select the end point for the arc.

■ Enter Radius (R) and specify a radius.
After entering the radius, the following prompt is displayed:

```
Specify arc end point or [Length]:
```

Select the end point or enter Length. If you use the Length option, specify a length and then a direction for the arc.

**NOTE** Because this option specifies a radius and the arc is tangent to the previous segment, the actual end point of the arc may not be at the selected end point.

■ Enter SecondPnt (S) and specify the second point for the arc.
After selecting the second point, the following prompt is displayed:

```
Specify arc end point or [Length]:
```

Select the end point or enter Length. If you use the Length option, specify a length and then a direction for the arc.

**NOTE** The Length option creates an arc that is tangent to the previous segment and passes though the second point. The arc may stop short of the second point if the length is shorter than the arc length to the point.

4 Specify the elevation of the end point.

5 Continue to select arc points or enter Line to draw straight segments or Close to close the feature line.

**To establish arc direction**

1 If you use the Arc option of the Draw Feature Line command prior to drawing any segments, it defaults to drawing a three-point arc. The following prompt is displayed:

```
Specify arc second point or [Direction]:
```

2 Do one of the following:
■ Click a location in the drawing to establish the direction of the arc.

■ Enter an angle at the command line. Use the DD.MMSS (degrees, minutes, seconds (page 2472)) format.

■ Enter Bearing (B). Temporary graphics are displayed on screen and the following prompt is displayed:

```
Specify quadrant (1-4) or [aZimuth/Angle]:
```

Use your mouse to click one of the four bearing quadrants as indicated by the temporary graphics, or enter the quadrant number at the command line.

After defining the quadrant, the following prompt is displayed:

```
Specify bearing or [aZimuth/Angle]:
```

Use your mouse to specify the bearing or enter it at the command line. Use the DD.MMSS format.

■ Enter Azimuth (Z). Temporary graphics are displayed on screen and the following prompt is displayed:

```
Specify azimuth or [Bearing/Angle]:
```

Use your mouse to specify the azimuth or enter it at the command line. Use the DD.MMSS format.

**3** After you have established the arc direction, the following prompt is displayed:

```
Specify arc end point or [Radius/SecondPnt/Line]:
```

See To draw curve segments of feature lines (page 750) for more information.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line 

Menu

Grading menu ➤ Draw Feature Line

Command Line

DrawFeatureLine

# Editing Feature Lines

You can use feature line editing commands with several different object types in addition to feature lines.

**NOTE** You can use the feature line commands to edit survey figures, including control points. You should update the survey database after editing a feature line. If you do not, the next time you recalculate the figure from the database, your edits to the feature lines will be overwritten.

**NOTE** You can also use the feature line commands to edit parcel lines.

The following table lists which objects are supported for each command.

| | Feature Lines | Parcel Lines | Survey Figures | 3D Polylines | 2D Polylines |
|---|---|---|---|---|---|
| Elevation Editor (page 755) | X | X | X | | |
| Quick Elevation Edit (page 754) | X | X | | | |
| Edit Elevations (page 757) | X | X | X | X | |

| | Feature Lines | Parcel Lines | Survey Figures | 3D Polylines | 2D Polylines |
|---|---|---|---|---|---|
| Set Grade/Slope Between Points (page 758) | X | X | X | X | |
| Set Elevation By Reference (page 759) | X | X | X | X | |
| Insert Elevation Point (page 765) | X | X | X | X<br><br>**NOTE** Inserts PI rather than elevation point. | |
| Delete Elevation Point (page 767) | X | X | X | | |
| Insert High/Low Elevation Point (page 767) | X | X | X | X | |
| Raise/Lower (page 764) | X | X | X | X | |
| Raise/Lower by Reference (page 761) | X | X | X | X | X |
| Adjacent Elevations by Reference (page 762) | X | X | X | X | |
| Grade Extension by Reference (page 763) | X | X | X | X | |
| Set Elevations From Surface (page 768) | X | X | X | X | |
| Insert PI (page 769) | X | X | X | X | X |
| Delete PI (page 771) | X | X | X | X | X |
| Join (page 771) | X | X | X | X | |
| Reverse (page 772) | X | X | X | X | X<br><br>**NOTE** This command also supports AutoCAD lines |

| | Feature Lines | Parcel Lines | Survey Figures | 3D Polylines | 2D Polylines |
|---|---|---|---|---|---|
| Edit Curve (page 773) | X | X | X | | |
| Fillet (page 776) | X | | X | X | |
| Fit Curve (page 777) | X | X | X | X | X |
| Smooth (page 779) | X | | X | | |
| Weed (page 781) | X | | | X | X |
| Stepped Offset (page 783) | X | | X | X | X |
| Apply Feature Line Names (page 746) | X | | | | |
| Apply Feature Line Styles (page 746) | X | | | | |
| Break (page 784) | X | | | | |
| Remove Dynamic Links (page 747) | X | | | | |
| Trim (page 785) | X | | | | |
| Adding a Feature Line to Surface as a Breakline (page 786) | X | | | | |
| Convert 2D to 3D Polyline (page 789) | | | | | X |
| Convert 3D to 2D Polyline (page 789) | | | | X | |
| Edit Polyline Elevations (page 790) | | | | | X |
| Quick Profile (page 1058) | X | X | X | X | X |

# Editing the Elevations of a Feature Line

You can edit the elevation of a feature line using the Grading Elevation Editor, the command line, or several different tools on the Edit Elevations panel.

You can use many of these commands on objects other than feature lines. See the Quick Reference tab of each topic for a list of supported objects for each command.

## Quickly Editing Feature Line Elevations

Use the Quick Elevation Edit command to interactively edit the elevation or grade of any feature line or parcel line in a specified site.

The elevation is displayed at vertices and elevation points, and the grade is displayed along segments. Click to edit the value on the command line.
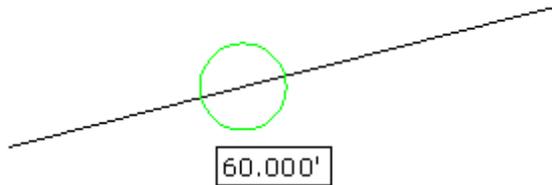


**To quickly edit feature line elevations**

1  Click Modify tab ➤ Design panel ➤ Elevations Drop-down ➤ Quick Elevation Edit .

   **NOTE** The current site is listed at the command line. If you want to select a different site or an object in the desired site, enter Site. Then, select either an object in the drawing or select Enter to access a list of available sites in the Select Object(s) dialog box.

2  Move your mouse over the geometry you want to edit.

   ■ Triangles  are displayed when you hover over a vertex or intersection point. The elevation is displayed in a tooltip.

   ■ Circles  are displayed when you hover over an elevation point (page 2473). The elevation is displayed in a tooltip.

   ■ Arrows  are displayed when you hover over a segment. The arrow points to the end of the segment that is closest to the cursor. This is the point that will have its elevation changed with the grade edit.The grade is displayed in a tooltip.

   ■ Gray glyphs  are displayed when you hover over an elevation or grade that cannot be edited. Uneditable lines include objects such as daylight and dynamically linked feature lines.

   When you locate the elevation or grade you want to edit, click your mouse.

3  Do one of the following:
   ■ Enter the new elevation or grade at the command line.

   ■ If you clicked an elevation point, you can also enter Surface to open the Select Surface dialog box and select a surface to obtain the elevation from.

   **NOTE** The Surface option only appears if there are surfaces in the drawing.

■ If you clicked a grade point, you can also enter Slope and specify a slope. It edits the elevation at the end of the segment in the direction that the arrow is pointing.

**NOTE** If the feature lines are being used as surface breaklines and the surface Rebuild-Automatic setting is turned on, you can see the surface updating automatically as you make each edit.

**To quickly edit feature line elevation points by selecting a reference**

1 Click Modify tab ➤ Design panel ➤ Elevations Drop-down ➤  Quick Elevation Edit.

2 Enter Reference.

3 Select the reference point to use. When picking the reference point, it defaults to a point in the site, but you can also select the Pick option to use OSNAP to pick any location in the drawing.

4 Move the cursor around the reference point and a yellow line is drawn on screen from the reference point to the closest point.

5 Do one of the following:
■ Enter a grade. The grade is applied between the reference point and the point on the feature line.

■ Enter Slope and then enter a slope. The slope is applied between the reference point and the point on the feature line.

■ Enter Difference and then enter the elevational difference. The elevational difference is applied between the reference point and the point on the feature line.

The assigned elevation for the point is displayed at the command line.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Quick Elevation Edit 

Menu

Click Grading menu ➤ Edit Grading Elevations ➤ Quick Elevation Edit

Command Line

QuickEditFeatureElevs

Supported Objects

■ Feature lines

■ Parcel lines

## Editing Feature Lines with the Grading Elevation Editor

Edit the vertex elevations of feature lines, survey figures, and parcel lines with the Grading Elevation Editor.

Use the Grading Elevation Editor to edit feature line elevations in a tabular dialog box. You can also assign an elevation from an existing surface.

Changes you make in the editor dynamically change the elevation in the drawing.

Within the editor, each vertex is displayed on its own row, and is marked with a triangle or circle.
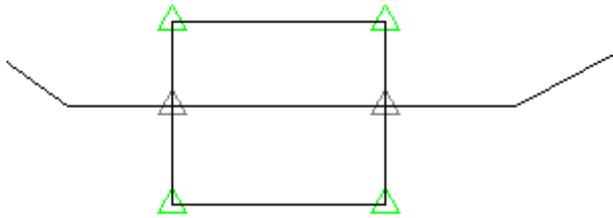
- Green triangles ▲ mark the points that represent the major horizontal geometry points. For example, when you create a feature line, all vertices are shown as triangles. If you insert points of intersection (PI), they are also shown as triangles.

- White triangles ▲ indicate split points, where two feature lines cross although neither has a geometry point at that location. You cannot directly edit the elevation of these points.

- Circles ● mark the elevation change points. Insert a new elevation point (page 2473) by using the Insert Elevation Point icon in the Grading Elevation Editor.

- Grade arrows display at segment midpoints (pointing downhill) to indicate feature line flow directions and to display the segment that is being edited when a single row is selected.

---

NOTE When a vertex is a shared point with another intersecting feature line, a small + symbol ▲ is displayed with the icon for that row.

---

When two features, such as feature lines, lot lines or survey figures cross and neither feature has a geometry point at that location, a split point is created. You cannot edit this point directly, as it is controlled by the grades of the two crossing segments which must have the same elevation at the crossing point. If the two elevations are different, a grade break is inserted into one to match the elevation of the other.

You can assign a priority to feature line styles so that when two feature lines with different styles cross, the one with the higher priority will set the elevation. Feature lines without a style have the lowest priority. Use the Options tab in the Feature Line Site Properties (page 1915) dialog box to set the feature line style priority.

If two feature lines have the same style or no style, then the feature line that is modified last becomes dominant, causing the other feature line to break at that point. To control the elevation at that point, use the Insert PI command to create a vertex at the split point.

When you click a row, a marker is displayed in the drawing, displaying the vertex that you are on.

**To edit a feature line, using the Grading Elevation Editor**

1 Click Modify tab ➤ Edit Elevations panel ➤ Elevation Editor ▦ .

2 Select the feature line or an other object.

3 Change the elevation in the Grading Elevation Editor (page 1889) dialog box.

The Elevations From Surface option is unavailable if there are no surfaces in the drawing. If there is only one surface, that surface is automatically used to assign the elevation. If there is more then one surface, you are prompted to select the desired surface.

If no feature line rows are selected, the elevation of all points are updated. If the feature line is off of the surface, a warning message displays. If some elevations are updated, but one or more points are off of the surface, a message displays, indicating the number of points that could not be assigned elevations.

The feature line is updated in the drawing, and the grading is adjusted.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Elevation Editor 

Menu

Click Grading menu ➤ Edit Figure Elevations ➤ Elevation Editor

Command Line

GradingElevEditor

Dialog Box

Grading Elevation Editor (page 1889)

Supported Objects

■ Feature lines

■ Parcel lines

■ Survey figures

## Editing Feature Line Elevations at the Command Line

Edits the vertex elevations of a survey figure, parcel line, or 3D polyline at the Command prompt.

Use this command to step through each vertex and elevation point on a feature line or figure. You can change elevations and grades, also insert, move, and delete elevation points.

**To edit a feature line from the command line**

1 Click Modify tab ➤ Edit Elevations panel ➤ Edit Elevations  .

2 Select the feature line or other object.

3 To select the vertex or elevation point to edit, press Enter without entering a value. You can also enter Next or Previous to change the direction the command uses to step through the points.

■ A triangle  is displayed on the feature line when you select a vertex or intersection point.

■ A circle  is displayed on the feature line when you select an elevation point (page 2473).

4 Do one of the following:
■ Enter an elevation and press Enter to modify the elevation and stay at the current point. The updated values are displayed.

■ Enter Grade and specify a grade.

■ Enter Surface and select a surface to obtain the elevation from.

- Enter Insert and select a location for a new elevation point along the feature line.
  The following options are available only for elevation points:

- Enter Move and select a new location for the elevation point.

- Enter Delete to delete the current elevation point.

5 As you make each change, the feature line is updated in the drawing and the grading is adjusted.

6 Enter Exit to end the command.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Edit Elevations

Menu

Click Grading  menu ➤ Elevation Editor ➤ Edit Elevations

Command Line

EditFeatureElevs

Supported Objects

- Feature lines

- Parcel lines

- Survey figures

- 3D polylines

## Setting a Grade or Slope Between Points

Use to edit the grade/slope between two vertices on a feature line, survey figure, parcel line, or 3D polyline

Specify grade, slope, point elevations, or elevation difference between points. For the selected segment (1), you specify the start point (2), the elevation (3), and the end point (4). Then specify the grade, slope, elevation or elevation difference.

**To set a grade or slope between points**

1 Click Modify tab ➤ Edit Elevations panel ➤ Set Grade/Slope Between Points ⬚ .

2 Select the feature line or other object.

3 Select the start point of the segment you want to edit.

4 Optionally, specify the elevation of the start point by entering an elevation or by entering Surface and selecting a surface to obtain the elevation from.

5 Select the end point of the segment.

6 Do one of the following to establish the grade or slope between the points:
   - Enter a grade.
   - Enter Slope and specify a slope.
   - Enter Elevation and specify an elevation for the end point.
   - Enter Difference and specify an elevational difference between the two points.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Set Grade/Slope Between Points ⬚

Menu

Click Grading menu ➤ Edit Figure Elevations ➤ Set Grade/Slope Between Points

Command Line

SetFeatureGrade

Supported Objects

- Feature lines
- Parcel lines
- Survey figures
- 3D polylines

## Setting Feature Line Elevation by Reference

Sets a vertex elevation on a feature line, survey figure, parcel line, or 3D polyline at a given grade/slope from a specified location

Example: Specify the elevation of a ditch footprint in relation to a building pad elevation. Temporary graphics show the reference point and the vertices on the feature line.

**To set feature line elevation by reference**

1 Click Modify tab ➤ Edit Elevations panel ➤ Set Elevation By Reference ⬚ .

2 Select the reference point to use.

The elevation of the selected point is displayed at the command line.

**3** Select the feature line or other object to which you want to apply the elevation.

Temporary graphics are drawn on-screen to show the reference point and the vertices and elevation points on the feature line.
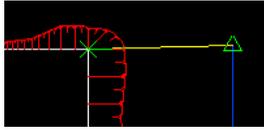


**4** Do one of the following:

- ■ Click to select the first point selected on the feature line. Or move your mouse to snap to a different point on the feature line, and then click to select it.

- ■ Enter Insert and then select a point on the feature line at which to insert a new elevation point. On 3D polylines, a new PI is inserted.

The distance between the selected point and the reference point, and the elevation and grade of the selected point, are displayed at the command line.

**5** Do one of the following:

- ■ Enter a grade. The grade is applied between the reference point and the point on the feature line.

- ■ Enter Slope and then enter a slope. The slope is applied between the reference point and the point on the feature line.

- ■ Enter Difference and enter the elevational difference. The elevational difference is applied between the reference point and the point on the feature line.

The assigned elevation for the point is displayed at the command line.

**6** The original reference point remains active. Select another feature line to continue setting elevations in reference to this point. Or, press Enter to end the command.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Set Elevation By Reference ⤢

Menu

Click Grading menu ➤ Edit Figure Elevations ➤ Set Elevation By Reference

Command Line

SetFeatureRefElev

Supported Objects

- ■ Feature lines

- ■ Parcel lines

- ■ Survey figures

- ■ 3D polylines

# Raising or Lowering Feature Line Elevation by Reference

Use the Raise/Lower By Reference command to raise or lower a feature line, based on a grade, slope or relative elevation from a reference point.

Select the Pick option to pick any location in the drawing or use the transparent commands to select a COGO point or surface elevation.

**To raise or lower feature line elevation by reference**

1   Click Modify tab ➤ Edit Elevations panel ➤ Raise/Lower By Reference ↗‡ .

2   Select the reference point to use. Select the Pick option to use OSNAP to pick any location, in the drawing or use the transparent commands to select a COGO point or surface elevation.

    The elevation of the selected point is displayed at the command line.

3   Select the feature line or other object that you want to raise or lower.

4   Select a point on the object you selected.

    The distance between the selected point and the reference point, and the elevation and grade of the selected point, are displayed at the command line.

5   Do one of the following:

    ■ Enter a grade. The grade is applied between the reference point and the point on the feature line.

    ■ Enter Slope and enter a slope. The slope is applied between the reference point and the point on the feature line.

    ■ Enter Difference, and enter the elevational difference. The elevational difference is applied between the reference point and the point on the feature line.

    The assigned elevation for the point is displayed at the command line.

6   The original reference point remains active. Select another feature line to continue setting elevations in reference to this point. Or, press Enter to end the command.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Raise/Lower By Reference ↗‡

Menu

Click Grading menu ➤ Edit Figure Elevations ➤ Raise/Lower By Reference

Command Line

SetFeatureRefElev

Supported Objects

■ Feature lines

■ Parcel lines

■ Survey figures

■ 3D polylines

■ Polylines

## Setting Adjacent Feature Line Elevations by Reference

Use the Adjacent Elevations By Reference command to set the elevations along a feature line, survey figure, lot line or 3D polyline, based on a grade, slope or elevation difference from points along another feature line.

This command is similar to setting elevations in stepped offset, without first creating the offset.

Using this command, you can edit the elevations of one feature line and then update the elevations of another related feature line that exists alongside the first.



**To set adjacent feature line elevations by reference**

1 Click Modify tab ➤ Edit Elevations panel ➤ Adjacent Elevations By Reference  .

2 Select the reference object.

3 Select the feature line or another object you want to edit.

Glyphs are displayed at each point that will be edited along the feature line, as well as yellow lines projected to the reference feature line, which display the location of the reference elevations for each point.
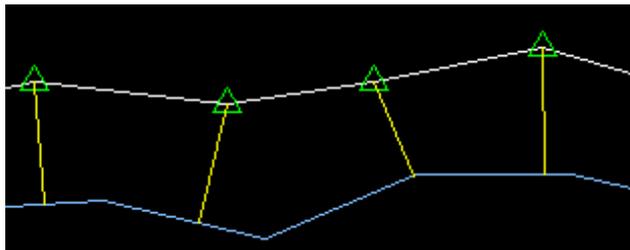


Only points that fall within the range of the reference object can be modified.

4 Do one of the following:

■ Enter the elevational difference. The elevational difference is applied between the reference point and the point on the feature line.

■ Enter Grade and enter a grade. The grade is applied between the reference point and the point on the feature line.

■ Enter Slope and enter a slope. The slope is applied between the reference point and the point on the feature line.

■ Enter Variable. The Variable option steps along each point prompting for the elevation difference, grade, or slope. It does not insert any new points along the feature

The assigned elevation for the point is displayed at the command line.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Adjacent Elevations By Reference

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Adjacent Elevations By Reference

Command Line

SetFeatureRefElev

Supported Objects

- Feature lines
- Parcel lines
- Survey figures
- 3D polylines

## Extending a Grading By Reference

Use the Grade Extension By Reference command to edit an elevation by extending the grade of a segment on another feature.

This command is used to apply the same grade from a segment across a gap to a point on another feature line. For example, you can extend a grade across an intersection.
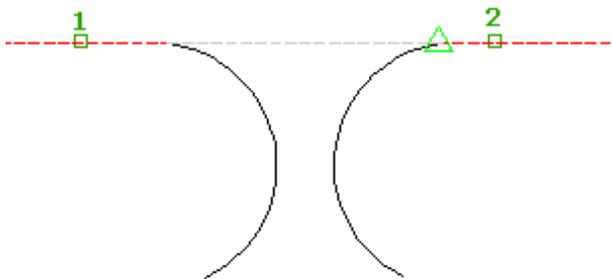


**To extend a grading by reference**

1   Click Modify tab ➤ Edit Elevations panel ➤ Grade Extension By Reference .

2   Select the reference segment.

    The segment is highlighted and the grade and slope are displayed in the command line.

3   Select the feature line or an other object that you want to edit.

    As the cursor is moved along the feature line, it snaps to the nearest point that you can edit, displaying a yellow line from the reference point to the edit point.
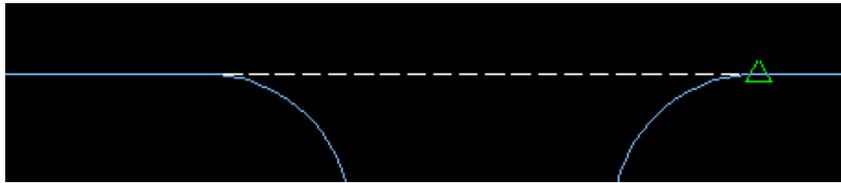
**4** Specify the point.

The grade, slope, elevation and distance to the point are displayed in the command line.

**5** Press Enter to use the default reference grade. Do one of the following:

- Enter a grade. The grade is applied between the reference point and the point on the feature line.

- Enter Slope and then enter a slope. The slope is applied between the reference point and the point on the feature line.

- Enter Difference and enter the elevational difference. The elevational difference is applied between the reference point and the point on the feature line.

The assigned elevation for the point is displayed at the command line.

**6** Enter the grade to that point. The reference grade is the default.

**7** Select another reference segment. Press Enter to end the command.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Grade Extension By Reference

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Grade Extension By Reference

Command Line

SetFeatureRefElev

Supported Objects

- Feature lines

- Parcel lines

- Survey figures

- 3D polylines

- Polylines

## Raising or Lowering Feature Lines

Use the Raise/Lower command to raise or lower the elevation for a selection of feature lines, figures, lot lines, polylines or 3d polylines.

If the selected feature lines all have the same constant elevation, an Elevation option is available to set the elevation value.

**To raise or lower features lines**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Raise/Lower $+\mathcal{I}^{\dagger}$ .

**2** Select the feature line or other objects to edit.

**3** Enter the elevation difference by which to raise or lower the feature lines.
The default elevation difference is 1.0.

**4** If the selected feature lines are all at the same elevation, an Elevation keyword appears. Select the Elevation option to specify an elevation for the feature lines. The default is the elevation of the feature lines.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Raise/Lower $+\mathcal{I}^{\dagger}$

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Raise/Lower

Command Line

RaiseLowerFeatures

Supported Objects

■ Feature lines

■ Parcel lines

■ Survey figures

■ 3D polylines

■ Polylines

## Inserting a Feature Line Elevation Point

Use the Insert Elevation Point command to insert an elevation point (page 2473) on a feature line.

Specify a location, distance, or increments at which to insert multiple points. When a single point is inserted, you can specify the elevation.
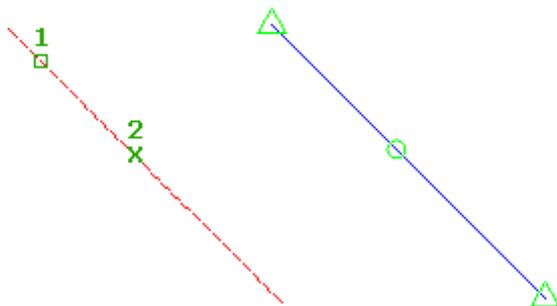
**To insert a feature line elevation point by selecting a location**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Insert Elevation Point ╪⬦ .

**2** Select the feature line or other object.

**3** Click to select the location along the feature line to insert the elevation point.

**4** Specify the elevation by entering a value. Or enter Surface to obtain the elevation from a surface. If only one surface is in the drawing it is selected automatically.

**5** Select another point or specify a distance to insert another point.

**To insert a feature line elevation point by specifying a distance**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Insert Elevation Point ╪⬦ .

**2** Select the feature line or other object.

**3** Enter Distance.

**4** Select an existing point on the feature line to specify the start of the distance. If you select a point other than the start or end point, you must then select the direction.

**5** Enter a distance to define the location of the elevation point.

**6** Do one of the following to define the elevation of the point:

- Enter a grade.

- Enter Slope and enter a slope.

- Enter Elevation and enter an elevation. Or enter Surface to obtain the elevation from a surface.

- Enter Difference and enter an elevational difference.

**To insert multiple elevation points at an increment**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Insert Elevation Point ╪⬦ .

**2** Select the feature line or other object.

**3** Enter Increment.

**4** Enter a value to define the length of the increment. Points are placed equidistant along the feature line using the specified increment.

**NOTE** Using this option, you are not prompted for elevations. The points are automatically assigned the elevation that exists at their location on the feature line.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Insert Elevation Point ╪⬦

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Insert Elevation Point

Command Line

InsertElevPoint

Supported Objects

- Feature lines

- Parcel lines

- Survey figures

- 3D polylines (inserts PI rather than elevation point)

## Deleting a Feature Line Elevation Point

Use the Delete Elevation Point command to delete an elevation point (page 2473).

Click to select the point to delete, or enter All to delete all elevation points.

**To delete a feature line elevation point**

1  Click Modify tab ➤ Edit Elevations panel ➤ Delete Elevation Point .

2  Select the feature line or other object.

3  Click to select the elevation point to delete. Or enter All to delete all elevation points.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Delete Elevation Point

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Delete Elevation Point

Command Line

DeleteElevPoint

Supported Objects

- Feature lines

- Parcel lines

- Survey figures

## Inserting a High or Low Elevation Point

Inserts a high or low elevation point onto a feature line, survey figure, parcel line, or 3D polyline.

This command inserts the break point where two grades intersect. Specify the grade or slope ahead from the start point and back from the end point.

**To insert a high or low feature line elevation point**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Insert High/Low Elevation Point  .

**2** Select the feature line or other object.

**3** Select the start point and the end point of the span that you want to add the point to. These points can span multiple feature line segments.

After you select the start and end points, an arrow is displayed at the start of the span. The command line displays the start and end elevations and the distance between the start and end points.

**4** Enter the grade (or enter Slope and specify a slope) from the start point.

Enter a positive value to grade up or a negative value to grade down.

**5** Enter the grade back from the end point.

**NOTE** If an intersection cannot be found with the entered values a "No solution found" message is displayed.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Insert High/Low Elevation Point 

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Insert High/Low Elevation Point

Command Line

InsertFeatureHighLowPoint

Supported Objects

■ Feature lines

■ Parcel lines

■ Survey figures

■ 3D polylines

## Setting Feature Line Elevations from a Surface

Use the Elevations From Surface command to assign the elevations of a surface to the geometry points of one or more feature lines, parcel lines, survey figures, or 3D polylines.

Elevations are assigned to each vertex. You can include intermediate elevation points at any break points along the surface. Elevations obtained from the surface are not dynamic.

**To set feature line elevations from a surface**

**1** Click Modify tab ➤ Edit Elevations panel ➤ Elevations From Surface  .

**2** In the Set Elevations From Surface (page 1914) dialog box, select a surface.

**3** Select the Insert Intermediate Grade Break Points check box to insert intermediate grade breaks where the entity crosses surface TIN lines. Elevation points are created at these locations.

**4** Click OK.

**5** Do one of the following:

- Select the feature line or other object.

- Enter Multiple, and then select multiple objects.

- Enter Partial, and then define a portion of the feature line to assign elevations to.

**6** The feature line is assigned the elevations of the underlying surface. To edit these elevations, see Editing the Elevations of a Feature Line (page 753).

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Elevations panel ➤ Elevations From Surface

Menu

Grading menu ➤ Edit Feature Line Elevations ➤ Elevations From Surface

Command Line

FeatureElevsFromSurf

Supported Objects

- Feature lines

- Parcel lines

- Survey figures

- 3D polylines

# Inserting Feature Line PIs

Use the Insert PI command to insert points of intersection on a feature line.

Points of intersection break the existing horizontal geometry of the feature line. You can use this command to convert a split point into a geometry point for editing.
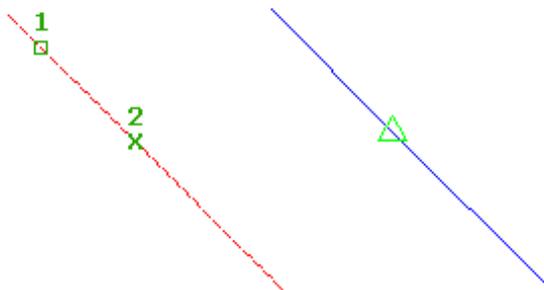


Split points are created when two feature lines cross and neither has a geometry point. A split point cannot be edited as it is controlled by the grades of the two crossing segments which must have the same elevation at the crossing point. If their elevations are different, a grade break will be inserted into one to match the

elevation of the other. If the feature lines have the same style or no styles are assigned, the segment that is modified last will become dominant, causing the other segment to break at that point. Use the Insert PI command to create a vertex at the split point so that you can have direct control over the elevation at that point.

**To insert points of intersection into feature lines**

1   Click Modify tab ➤ Edit Geometry panel ➤ Insert PI ⊹.

2   Select the feature line or other object.
    Temporary graphics are drawn on screen.
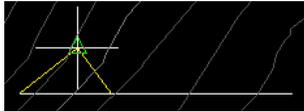
    

3   Click to insert the PI.

4   Enter Distance. Then specify a distance from the start point at which a PI point along the feature line is inserted.

5   Enter Increment. Specify distance between points, then click the feature line to to automatically insert PI points separated by the specified distance. As this command inserts multiple points, the elevation for each point will be determined by the existing elevation of the feature line at each point.

6   Do one of the following:
    ■   Enter the elevation for the point.

    ■   Enter Surface to obtain the elevation from a surface in the drawing.

    **NOTE** You are not prompted for the elevation if the object you selected is a 3D polyline.

**To insert points of intersection into a crossing point**

1   Click Modify tab ➤ Edit Geometry panel ➤ Insert PI ⊹.

2   Use OSNAP to select the intersection point.
    Temporary graphics are drawn on screen.

3   Click to insert the PI.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Insert PI ⊹

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Insert PI

Command Line

InsertFeaturePI

Supported Objects

■   Feature lines

- Survey figures
- 3D polylines
- 2D polylines

## Deleting Feature Line PIs

Deletes a selected vertex on a feature line, survey figure, polyline, or 3D polyline.

Use this command to delete points of intersection from feature lines.

**To delete points of intersection from feature lines**

1 Click Modify tab ➤ Edit Geometry panel ➤ Delete PI ✖️.

2 Click the feature line or other object near the PI you want to delete.

### Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Delete PI ✖️

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Delete PI

Command Line

DeleteFeaturePI

Supported Objects

- Feature lines
- Survey figures
- 3D polylines
- 2D polylines

## Joining Feature Lines

Use the Join command to join connecting feature lines, polylines, or 3D polylines into one feature line or survey figure.

Select the two lines. They are joined if they lie within the tolerance distance set in JoinFeatures (page 736) command settings.
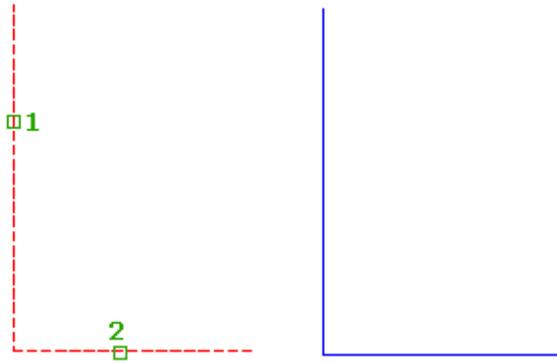
**To join feature lines**

1 Click Modify tab ➤ Edit Geometry panel ➤ Join ⤬.

2 Select the feature line you want to join other lines to.

3 Select the joining object or enter Multiple to make a selection of multiple feature lines, polylines, or 3D polylines.

## Quick Reference

Menu

Click Modify tab ➤ Edit Geometry panel ➤ Join ⤬.

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Join.

Command Line

JoinFeatures

Supported Objects

■ Feature lines

■ Survey figures

■ 2D polylines

■ 3D polylines

# Reversing Feature Lines

Use the Reverse command to change the direction of feature lines.

This command affects labeling and stationing of feature lines.
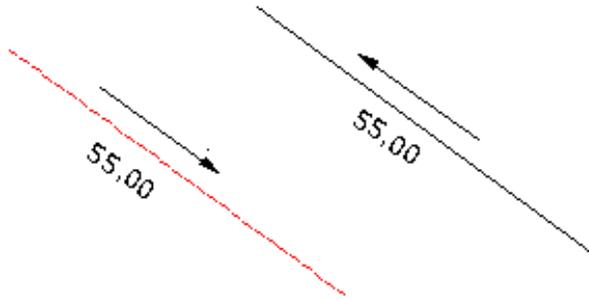
**To reverse feature lines**

1  Click Modify tab ➤ Edit Geometry panel ➤ Reverse  .

2  Select the feature line or other object.

3  Select additional feature lines that you want to reverse. Or, press Enter to end the command.

> **NOTE** Use Zoom to see the directional arrows that display on each segment.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Reverse 

Menu

Grading menu  ➤  Edit Feature Line Geometry  ➤  Reverse

Command Line

ReverseFeature

Supported Objects

■  Feature lines

■  Survey figures

■  3D polylines

■  2D polylines

■  Lines

# Editing Feature Line Curves

Use the Edit Curve command to edit the radius of feature line (or survey figure) arcs.

When the radius is changed, the new arc is displayed and the existing arc is indicated with dashed highlighting.
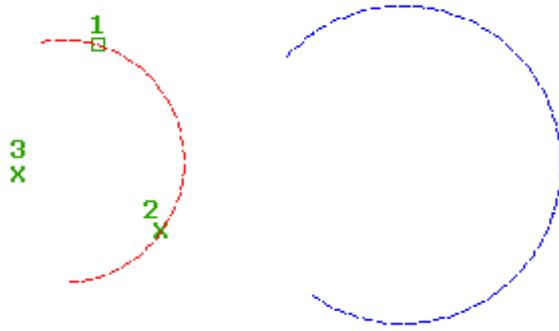
**To edit feature line curves**

1 Click Modify tab ➤ Edit Geometry panel ➤ Edit Curve  .

2 Select the curve to edit. The curve is highlighted, and the points are displayed.

3 In the Edit Feature Line Curve (page 1916) dialog box, edit the curve. Use the Radius edit box to specify the radius.

4 Use the Maintain Tangency option to control the curve tangency state.
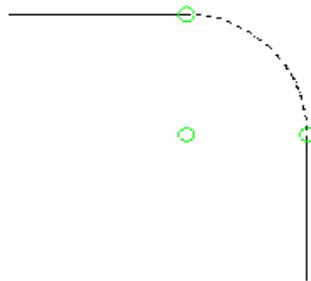


**The curve is highlighted and the arc start, center and points are displayed with glyphs.**

When the radius is changed, if the arc is tangent to the segments on either side, the tangency is maintained. Before selecting OK in the Edit Feature Line Curve dialog box, the new arc is displayed. At the same time, the existing arc remains visible with the dashed highlighting.
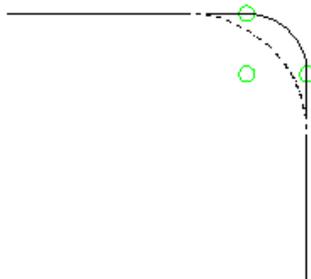


**The start and end points remain constant if the arc is not tangent.**

If the arc is not tangent, the start and end points of the arc are held when the radius is changed.
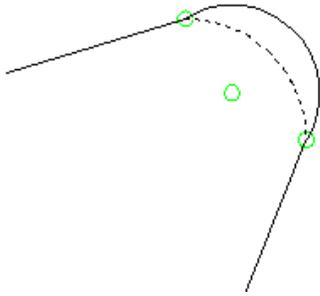
If the curve is not tangent, the Maintain Tangency button is activated. If you toggle this button the curve will be adjusted to make it tangent to the adjacent segments. If the curve is tangent, the button is disabled.
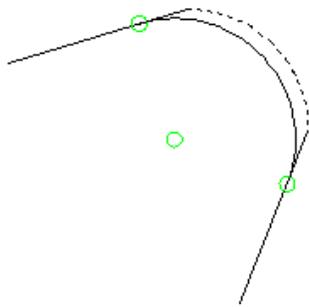
**NOTE** When the Delete option is selected, the segments on either side of the curve are extended to an intersection point.

**Segments extend to intersection point.**
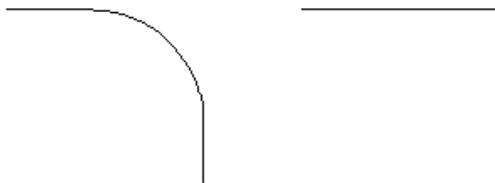
## Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Edit Curve

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Edit Curve

Command Line

EditFeatureCurve

Supported Objects

- Feature lines
- Survey figures
- 3D polylines
- 2D polylines
- Lines

# Filleting Feature Lines

Fillets a feature line, survey figure, or 3D polyline.

This command can be used to round the corners of feature lines. The fillet can span multiple segments and can be used to join two connecting feature lines.
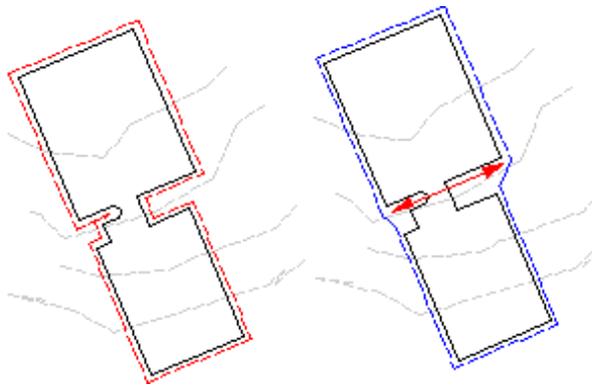


**To fillet feature lines**

1   Click Modify tab ➤ Edit Geometry panel ➤ Fillet ✛ .

2   Select the feature line or other object.

The currently specified radius is displayed at the command line.

3   Do one of the following:

- Select the corner to fillet. When you move your mouse over the feature line, a triangle highlights the corners that can be filleted and the current radius value is displayed.

   **Single corner fillet**
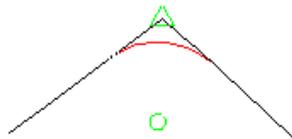


   **Multiple corner fillet**
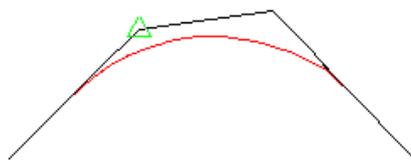
- Enter Radius to specify a different radius.

- Enter All to fillet all corners.

- Enter Join to join a connecting feature line and then select the feature line.

### Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Fillet ⊹

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Fillet

Command Line

FilletFeature

Supported Objects

- Feature lines

- Survey figures

- 3D polylines

# Converting Tessellated Lines to Arcs

Fits a curve from a selection of vertices with a feature line, survey figure, parcel, or 3D polyline

Use this command to convert tessellated lines to true arcs, or to add fillets to straight segments. You can specify start and end points or set options for automatic arc generation.
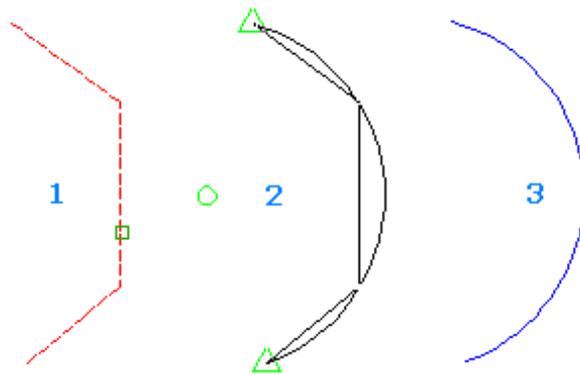


Tessellation occurs when you convert a 2D polyline with arcs to a 3D polyline. It also occurs when you export feature lines with arcs from corridor models.

**Grading created from tessellated footprint (left) compared with grading created from arc (right)**
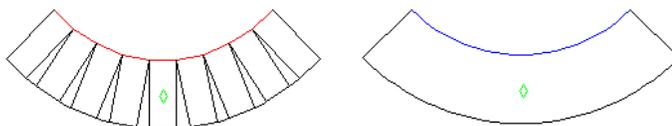
Feature line points that define a grade break are inserted as elevation points on the arc when using this command.

There are three ways to select the segments to turn into arcs.

- **Automatic selection**: As you move your mouse over the feature line, arcs appear where they can be inserted. Click your mouse to insert the arc.

- **Specifying start and end points over multiple segments**: Use this method to define an arc using specified start and end points.

- **Specifying start and end points for a single segment**: Use this method to replace a single segment (such as a chamfered corner) with an arc. If the corner is asymmetrical, two arcs are inserted that are tangent to the line segments and to each other.
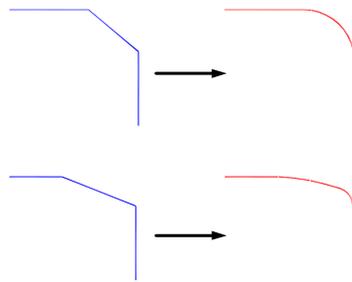


You can specify the command options by editing the FitCurveFeature command settings, or by using the Options keyword when running the Fit Curve command.

**To fit a curve**

1  Click Modify tab ➤ Edit Geometry panel ➤ Fit Curve  .

2  Select the feature line or other object.

3  Do one of the following:

- Move your mouse over the feature line. Arcs appear where they can be inserted. Click your mouse to insert an arc.

- Enter Points (P). Then select the start and end point on the feature line for the arc location. This option ignores the minimum number of segments setting.
  If the start and end points span more than one segment but the points do not define an arc, an error message is displayed indicating that an arc cannot be created between the specified points. This error occurs if the segments do not all curve in the same direction or if there is a straight segment that is composed of multiple small segments (which therefore have no deflection angle).

  **NOTE**  If the segments can define an arc but are not within the tolerance, you are prompted with an option to override the tolerance setting and insert the arc.

- Enter Options (O) to display the Fit Curve (page 1913) dialog box where you can change the tolerance and minimum number of segments settings.

**Quick Reference**

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Fit Curve ⌒

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Fit Curve

Command Line

FitCurveFeature

Supported Objects

- Feature lines
- Survey figures
- 3D polylines
- 2D polylines

# Smoothing Feature Lines

Use to replace feature line segments with arcs.

Use this command to smooth feature lines that were created from tessellated polylines. Select the Straighten option command to restore straight lines.



**Before and after smoothing**

When you select the object after smoothing, red tangent lines extend from the grips at the ends of the feature. You may need to zoom into the end to see them.

**Zoom in to end of feature line to see grip**
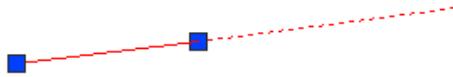
You can use these grips to adjust the arc directions at the start and end of the feature line.

**Select and move grip to adjust arc direction**
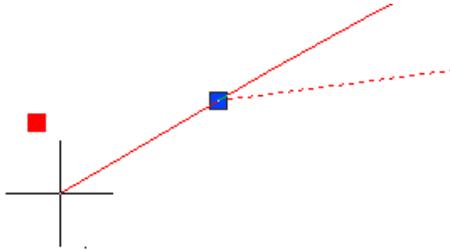
**Zoomed out view showing arc adjustment**

**To smooth feature lines**

1　Click Modify tab ➤ Edit Geometry panel ➤ Smooth ⌇ .

2　Select the feature line(s) to smooth or straighten.

3　Do one of the following:

■　Press Enter to smooth the lines.

■　Enter Straighten to straighten lines that were previously smoothed.

## Quick Reference

Ribbon

　Modify tab ➤ Edit Geometry panel ➤ Smooth ⌇

Menu

　Grading menu  ➤  Edit Feature Line Geometry  ➤  Smooth

Command Line

　SmoothFeature

Supported Objects

■ Feature lines

# Weeding Vertices

Use the Weed Vertices command to reduce the number of vertices on a feature line.

Vertices are weeded if found to be unnecessary for line location, based on weeding factors of angle, grade, length, and 3D distance. Points selected for weeding are highlighted in red.
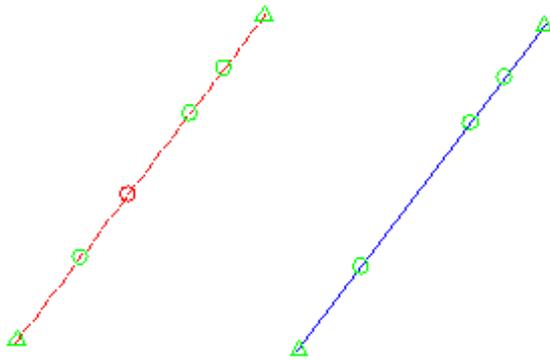


The weeding factors delete vertices that:

■ Are closer together in 2D space to their two neighboring vertices than the Length factor.

weeding length factor (plan view)

possible vertex to be deleted

L
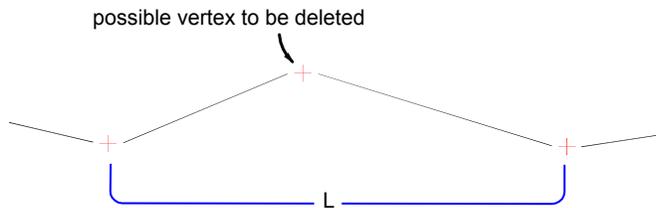
if L < length factor, then the vertex will be deleted

■ Have a 2D deflection angle that is less than the Angle factor.

weeding angle factor (plan view)

possible vertex to be deleted
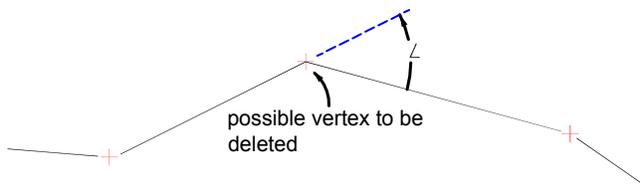
if angle < angle factor, then the vertex will be deleted

■ Have a difference in grade that is less than the Grade factor.

weeding grade factor (profile view)



possible vertex to be deleted
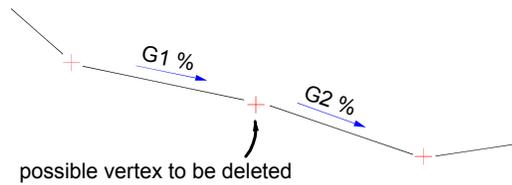
if the difference between G1 and G2 < grade factor,
then the vertex will be deleted

■ Are closer together in 3D space to their two neighboring vertices than the 3D Distance factor.

weeding 3D distance factor (profile view)



possible vertex to be deleted
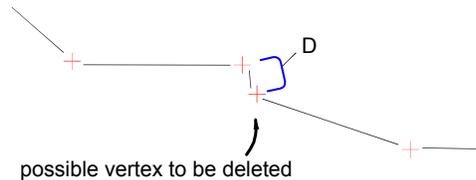
if D is < 3D distance factor, then the vertex will be deleted

Larger factors weed more vertices.

You can choose which factors you want to be used to calculate the weeding. For example, if you only want to apply a length factor, select the Length check box and specify a length, and clear all other check boxes.

You can set the defaults to be used for this command by editing the WeedFeatures command settings.

**To weed vertices**

1 Click Modify tab ➤ Edit Geometry panel ➤ Weed  .

2 Do one of the following:
   ■ Select the feature line or other object.

   ■ Enter Multiple and then select multiple objects to weed.

   ■ Enter Partial and then select the object and the portion of the object to weed.

   The vertices of the object(s) you selected are highlighted with green triangles and the Weed Vertices (page 1911) dialog box is displayed.

3 Select the check boxes for the weeding factors that you want to apply and specify the values to use.

When you specify a value that will delete a vertex, that vertex is highlighted in red in the drawing, and a message is displayed in the dialog box indicating how many vertices will be deleted. After you change a value, press Tab to update the message and the drawing display.

**Points selected for weeding are highlighted in red**
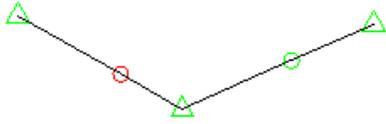


4 Click OK to delete the vertices.

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤  Weed

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Weed

Command Line

WeedFeatures

Supported Objects

■ Feature lines

■ 2D polylines

■ 3D polylines

# Creating a Stepped Offset

Creates a new feature line from an offset and difference in elevation from a selected feature line, survey figure, polyline, or 3D polyline.

Specify the offset distance, the side, and elevational difference or absolute elevation.
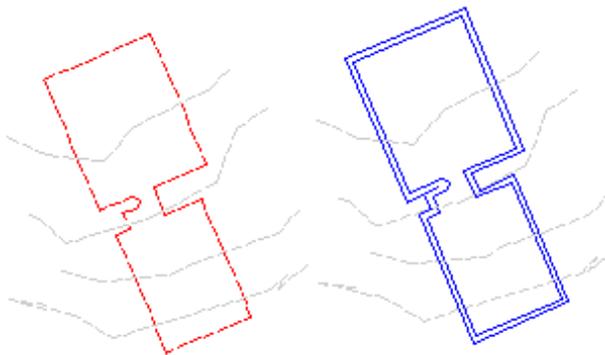
**To create a stepped offset by specifying a distance**

1 Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line From Stepped Offset ⟳ .

2 Do one of the following to specify the location for the offset:

■ Specify the offset distance, then select the object to offset.

■ Enter Through, then select the object to offset and select a through point. The offset will be drawn through this point.

3 Do one of the following:

■ Specify a point on the side where you want to place the offset. Then specify the elevational difference or the absolute elevation for the offset.

■ Enter Multiple and specify the side and elevational difference for each offset.

If you enter Variable, it will display a marker at the first point on the offset feature line. You can specify the elevation difference from the original point or the absolute elevation. After you enter a value, it will move to the next point and prompt for its elevation. It will continue to do this for each point on the feature line until you end the command.

4 Select another object to offset, or press Enter to end the command.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line From Stepped Offset ⟳

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Stepped Offset

Command Line

OffsetFeature

Supported Objects

■ Feature lines

■ Survey figures

■ 3D polylines

■ 2D polylines

# Breaking a Feature Line

Use the Break command to break a feature line into two feature lines.

The prompts for this command are similar to the AutoCAD Break command.

Click two points to specify both sides of the break. By default, the point at which you select the feature line is the first break point, or enter First to select a different break point.
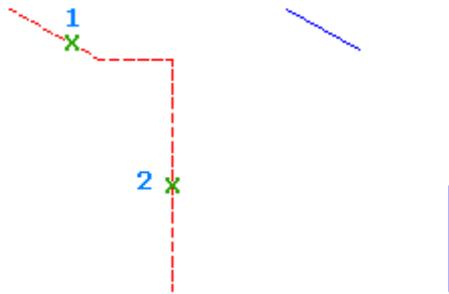
**To break a feature line**

1  Click Modify tab ➤ Edit Geometry panel ➤ Break ⊹ .

2  Select the feature line to break:

3  Specify the second break point.

---

**NOTE** if you enter the @ character at the prompt for the second break point, it uses the first point location, creating a break without a gap.

---

## Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Break ⊹

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Break

Command Line

Break

Supported Objects

■  Feature lines

# Trimming a Feature Line

Use this command to trim feature lines.

Use this command to trim feature lines so that they end precisely at boundary edges defined by other objects.
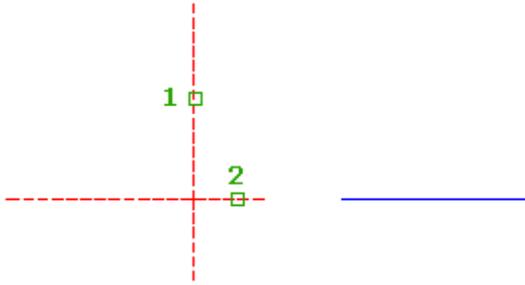
**To trim a feature line**

**1** Click Modify tab ➤ Edit Geometry panel ➤ Trim ⟋.

**2** Select the objects to serve as cutting edges.

**3** Select the feature line(s) to trim.

### Quick Reference

Ribbon

Click Modify tab ➤ Edit Geometry panel ➤ Trim ⟋.

Menu

Grading menu ➤ Edit Feature Line Geometry ➤ Trim

Command Line

Trim

Supported Objects

■ Feature lines

## Adding a Feature Line to a Surface as a Breakline

Adds feature lines to a surface as breakline data.

Use this command to select one or more feature lines from a drawing and add them to a surface. You can use breaklines to define features, such as retaining walls, curbs, tops of ridges, and streams.

When this command is selected, it will prompt for the surface, then prompt with the Add Breaklines dialog box.

**To add a feature line to a surface as breakline**

**1** Right-click a feature line ➤ Add To Surface As Breakline.

> **NOTE** If there are no surfaces in the drawing, you are prompted to create one.

**2** In the Select Surface dialog box, select a surface, and press OK. Or, select the Add button to create a new surface.

**3** In the Add Breaklines (page 2360) dialog box, specify the breakline options. Press OK.

> **NOTE** If you list a feature line or view its properties, you will see the names of the surfaces and breakline collections that the feature line is part of.

### Quick Reference

Ribbon

Select a feature line. Click Feature Line tab ➤ Modify panel ➤ Add To Surface As Breakline 

Toolspace Shortcut Menu

Right-click feature line ➤ Add to Surface as Breakline

# Labeling Feature Lines

Creates line or curve labels on a feature line or survey figure.

You can label single and multiple segments of feature lines by using specified line and curve label styles.
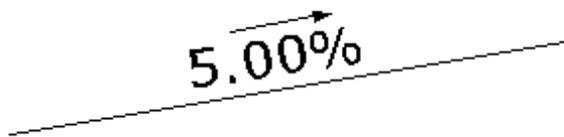


On the Toolspace Settings tab, in the General Label Style collection, set up Line and Curve label styles (page 1479) to use for the feature lines.

On the Toolspace Settings tab, in the General Label Style collection, specify the default line label style (page 1476).

Use the Single Segment or Multiple Segment command to quickly label a feature line using the default label style.

Use the Add Feature Line Labels option to access the Add Labels (page 1970) dialog box if you need to create or modify a style before labeling a feature line.

# Labeling Individual Feature Line Segments

Creates line or curve labels on a selected feature line or survey figure segment.

You can label single feature line segments, one at a time, with specified Line and Curve label styles.

**To label individual feature line segments**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Feature Line ➤ Single Segment 
.

**2**   Select the feature line or segment to label.

OR

**1**   Click Annotate tab ➤ Add Feature Line Labels ➤ Labels & Tables panel ➤ Feature Line ➤ Add Feature Line Labels

**2**   In the Add Labels dialog box (page 1970), under Feature, ensure that Line And Curve is selected.

**3**   Under Label Type, select Single Segment.

**4**   Specify the other options as required.

**5**   Click Add and then select the feature line or segment to label.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Feature Line ➤ Single Segment

Menu

Grading ➤ Add Feature Line Labels ➤ Single Segment
Grading ➤ Add Feature Line Labels ➤ Add Feature Line Labels

## Labeling Multiple Feature Line Segments

Creates line or curve labels on multiple line or curve objects, feature lines, or survey figure segments.

You can label multiple feature line segments with specified Line and Curve label styles.

**To label multiple feature line segments**

**1**   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Feature Line ➤ Multiple Segments .

**2**   Select the feature lines or segments to label.

OR

**1**   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels button .

**2**   In the Add Labels dialog box (page 1970), under Feature, ensure that Line And Curve is selected.

**3**   Under Label Type, select Multiple Segment.

**4**   Specify the other options as required.

**5**   Click Add and then select the feature lines or segments to label.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Feature Line ➤ Multiple Segments
Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels button

Menu

Grading ➤ Add Feature Line Labels ➤ Multiple Segments
Grading ➤ Add Feature Line Labels ➤ Add Feature Line Labels

# Editing Polylines

You can use the Polyline Utilities to convert between types of polylines and to assign elevations to 2D polylines.

## Converting 2D Polylines to 3D

Use the Convert 2D To 3D Polylines command to convert 2D to 3D polylines.

Use this utility if you need to modify the elevations of individual vertices of a 2D polyline. 2D polylines are coplanar objects. You can assign elevations to 3D polylines with the feature line elevation commands.

**To convert 2D polylines to 3D polylines**

1  Click Modify tab ➤ Design panel ➤ 🔻 ➤ Convert 2D To 3D Polylines 🔧.

2  Select the polyline(s) to convert. Press Enter.

### Quick Reference

Ribbon

Modify tab ➤ Design panel ➤ 🔻 ➤ Convert 2D To 3D Polylines 🔧

Menu

Grading menu ➤ Polyline Utilities ➤ Convert 2D To 3D Polylines

Command Line

ConvertPlines

Supported Objects

■  2D polylines

## Converting 3D Polylines to 2D

Use the Convert 3D To 2D Polylines command to convert 3D to 2D polylines.

The new polyline will take on the elevation of the start point of the 3D polyline.

**To convert 3D polylines to 2D polylines**

1  Click Modify tab ➤ Design panel ➤ 🔻 ➤ Convert 3D To 2D Polylines 🔧.

2  Select the polyline(s) to convert. Press Enter.

### Quick Reference

## Editing Polyline Elevations

Use the Edit Polyline Elevations to specify an elevation for a 2D polyline.

This command is useful for editing polylines that are being used as contours.

**To edit polyline elevations**

1  Click Modify tab ➤ Design panel ➤ ▼ ➤ Edit Polyline Elevations ⏣ .

2  Select the polyline.

3  Specify the elevation. All vertices are assigned the same elevation.

### Quick Reference

## Creating Grading

You can use the Grading Creation Tools to create grading from a footprint or to copy the criteria and style from another grading.

Use the tools to specify the grading group and criteria and then use the Create Grading options.

# Creating Grading from a Footprint

Before you create grading, you need to select a grading group to associate the grading objects with. After you create a grading object, you can move it to a different grading group within the same site.

Feature lines and grading are created by default on the layers specified by the drawing settings (page 1854).

Use the Grading Creation Tools (page 1892)to specify the grading criteria and style for the grading. Then create the grading by selecting the footprint from which the grading is projected. Command prompts vary, depending on the values that have been set in the grading criteria. For example, if the criteria has a target of Distance, which is not locked (page 739), you are prompted to enter a number for the distance.

**See also:**

■ Changing Grading Command Settings (page 736)

**To create grading from a footprint**

1   Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools .

2   Click and specify the grading group. Or create a new group.

3   Click to override the layer setting and then specify the layer for the grading.

4   Select a grading criteria for the grading from the list.

    If you have more than one set of named grading criteria, click to select the grading criteria set you want.

5   If the criteria you selected uses Surface as its target, specify which surface to target. The selected surface is displayed in the status bar of the layout toolbar. To select a different surface, click .

6   Select a grading style for the grading to use from the drop-down list.

    If you don't see the style selector, click to display the list of styles.

7   Click the arrow and then click Create Grading.

8   Click the footprint in the drawing. If you select a line, arc, or polyline, you are prompted to convert it to a feature line.

9   Click in the drawing to indicate the side of the footprint that you want to apply the grading. If you are grading from another grading's daylight line, you are not prompted for the grading side because only one side is available.

10  Specify whether the grading should be applied to the entire length of the line. To apply the grading to a subset of the line, enter n, then follow the command line prompts and on-screen arrows to specify the start and end points on the footprint.

    Depending on the criteria definition, you may be prompted to specify a distance and a slope value. For more information about the various options, see Criteria Tab (Grading Criteria Dialog Box) (page 1883).

11  Respond to any remaining prompts to complete the grading.

**Quick Reference**

Ribbon

Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools 

Menu

Grading menu ➤ Create Grading

Command Line

CreateGrading

Dialog Box

Grading Creation Tools (page 1892)

# Creating Grading by Copying an Existing Grading

Copy the criteria and style from an existing grading and apply it to a grading you create.

After selecting the grading from which to copy the criteria and style, create the new grading by selecting the footprint from which the grading is projected and selecting the side of the footprint from which to grade.

**To create grading by copying an existing grading**

1   Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools  .

2   In the Grading Creation Tools (page 1892) dialog box, click the arrow next to  . Click  Copy Create Grading.

3   In the drawing, select the grading from which you want to copy the criteria and style.

4   Select the footprint in the drawing and select the side of the footprint from which to grade.

   If you are grading from another grading's daylight line, you are not prompted for the grading side because only one side is available.

5   Click in the drawing to indicate the side of the footprint to which you want to apply the grading.

   Depending on the criteria definition, you may have to specify a distance and a slope value. For more information about the various options, see Criteria Tab (Grading Criteria Dialog Box) (page 1883).

6   Specify whether you want to apply the grading to the entire length of the line. To apply the grading only to a region, enter n and then specify the start and end points on the footprint that define the region.

7   Respond to any remaining prompts to complete the grading.

**Quick Reference**

Ribbon

Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools 

Menu

Grading menu ➤ Create Grading

Command Line

CopyCreateGrading

Dialog Box

Grading Creation Tools (page 1892)

# Editing Grading

You can edit either grading criteria or the elevations of the footprint or daylight line using edit commands on the Grading menu or the Grading Creation Tools.

Use the Grading Editor (Panorama) to edit unlocked criteria values for a grading object, the Grading Elevation Editor (Panorama) for feature lines and parcel lines, and additional commands for specific editing operations. You can also use grips to edit grading.

# Editing Grading Criteria

Edit a grading object with the Grading Editor vista.

Use the Grading Editor to change the unlocked criteria values for a grading object.

You can make changes to a sequence of grading objects, selecting them one after another with a single click. You can also edit a single grading object with the Edit Grading (page 794) command, by entering the new criteria values on the command line.

**To edit grading criteria**

1 Select the grading. Click Grading tab ➤ Modify panel ➤ Grading Editor ▦ .

2 In the drawing, select a grading.

The name of the criteria that is applied to the grading appears in the dialog box.

3 Edit the criteria for the grading.

For more information about the values that you can set for grading criteria, see Criteria Tab (Grading Criteria Dialog Box) (page 1883).

## Quick Reference

Ribbon

Select the grading. Grading tab ➤ Modify panel ➤ Grading Editor ▦

Menu

Grading menu ➤ Edit Grading ➤ Grading Editor

Toolspace Shortcut Menu

Settings tab: Right-click Grading Criteria item ➤ Edit

Command Line

GradingEditor

Dialog Box

Grading Criteria (page 1882)

# Editing Grading Criteria (Command Line)

Changes unlocked criteria values for a selected grading object using the command line.

Use the Grading Editor (page 793) to edit a sequence of grading objects by selecting them one after another with a single click.

**To edit grading criteria on the command line**

1   Select the grading. Click Grading tab ➤ Modify panel ➤ Edit Grading 🖊 .

2   In the drawing, select a grading.

3   In response to the prompts, edit the criteria for the grading.

   The prompts that you see for a particular grading object depend on the criteria that the grading currently uses. For more information about the values that you can set for grading criteria, see Criteria Tab (Grading Criteria Dialog Box) (page 1883).

## Quick Reference

Ribbon

   Select the grading. Grading tab ➤ Modify panel ➤ Edit Grading 🖊

Menu

   Grading menu ➤ Edit Grading ➤ Edit Grading

Command Line

   EditGrading

# Inserting a Transition

Add a transition to fill an area between two existing grading objects or to create a grading that has transitioning values.

A transition is an area that merges the grading of the regions on either side, for example, from 3:1 to 2:1.



There are several ways to create transitions:

■  You can select a location within an existing grading.

existing grading
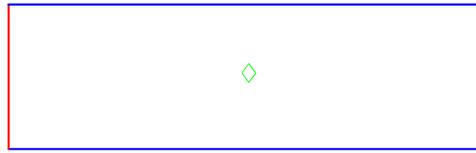
transition inserted
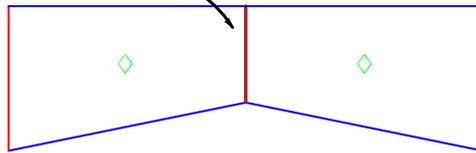
■ You can select a location between two existing gradings. The transition start and end points are defined by the existing projection lines.



existing grading
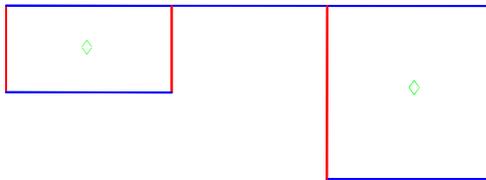
transition inserted
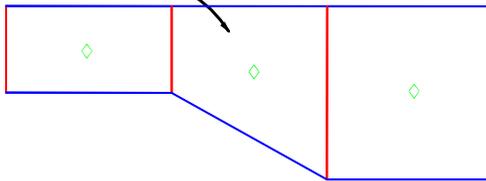
■ You can select start and end points for the transition and then define the parameters to use at each end. You can use the same criteria for both ends or you can switch criteria.



In addition, you are sometimes prompted to create transitions when you use the Create Grading command. This occurs when the start point of a new grading is inside or touching an existing grading.

new grading insertion point

grading inserted and transitions created
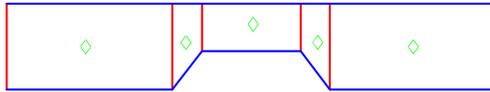
**To insert a transition between existing gradings**

1  Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools.

2  Click the arrow next to. Click Create Transition.

3  Select the feature line.

4  Click the location where you want to add the transition (between two existing regions).
   The transition is added and the grading is updated.

**To insert a transition within an existing grading**

1  Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools.

2  Click the arrow next to. Click Create Transition.

3  Select the feature line.

4  Enter Points.

5  Select the grading side.

6  Select the location where you want to add the transition (within an existing grading). The station of the point you selected is displayed at the command line. You can change it or press Enter to accept the location.
   The transition is added and the grading is updated.

**To insert a transition by defining different criteria and values**

1  Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools.

2  Click the arrow next to. Click Create Transition.

3  Select the feature line.

4  Enter Points.

5  Select the grading side.

6  Select the location where you want to add the transition. The station of the point you selected is displayed at the command line. You can change it or press Enter to accept the location.

7  Follow the prompts to define the criteria values for the start of the transition.

After you define the values, the following prompt is displayed:

```
Select the end point or [Criteria/Next/Previous/Both/Length]:
```

**8** Do one of the following:

■ Select the end point by clicking a location in the drawing. Then follow the prompts to define the criteria values for the end point.

■ Enter Length to specify a length for the transition. Then follow the prompts to define the criteria values for the end point.

■ Enter Criteria and select a different criteria to apply to the end point of the transition. If you select a criteria that is different from the criteria applied to the start point, the two are blended to create the transition.

---

**NOTE** The following three options are available only if other gradings exist on the footprint.

---

■ Enter Next to define the transition end point as the edge of the next grading on the footprint (up station).

■ Enter Previous to define the transition end as the edge of the previous grading on the footprint (down station).

■ Enter Both to straddle the grading transition between the next and previous grading edges on the footprint. This option overrides the selected start point.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools 

Command Line

CreateGradingTransition

Dialog Box

Grading Creation Tools (page 1892)

# Filling a Void Area

Fill a hole in the site grading by creating a grading with no criteria applied to it.

Any area bounded by feature lines or lot lines that is not already a grading should be converted to an infill grading to ensure correct contour display and volume calculations.
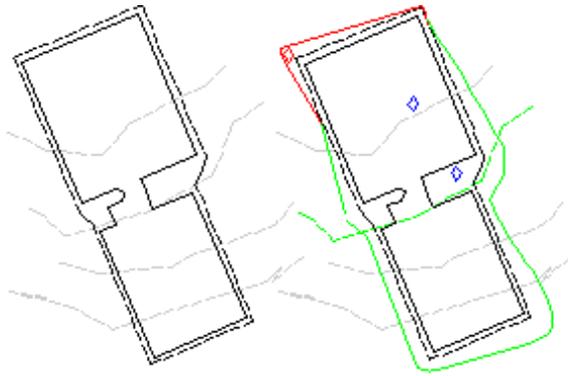
**To fill a void area**

1 Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools ⬛.

2 Click the arrow next to ⬛. Click ⬛ Create Infill.

3 Click the area that you want to fill. Valid areas that can be converted to infill are highlighted as you move the cursor over them.

The grading face is added and the grading is updated.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Grading drop-down ➤ Grading Creation Tools ⬛

Command Line

CreateGradingInfill

Dialog Box

Grading Creation Tools (page 1892)

# Editing with Grips

Use grips to adjust the component lines in a grading by changing the footprint geometry or the grading components.

Click a projection line in a region. Then change the start or end station point for that region by using a grip to drag the line to another location.

| Component | Description |
| --- | --- |
| Feature lines | Grip appears at each vertex. Move the grip as you would when you edit a polyline. |
| Projection lines | Grip appears on projection lines. Slide the projection line along the footprint to change the start or end point of the region. |
| Elevation points | Grip appears at any location on a feature line or lot line to mark an intermediate elevation point. Move the point to its correct location. |

**To edit using grips**

**1** In the drawing, select the grading or feature line you want to edit.

**2** Click the grip. The grip turns red to show it is active

**3** Drag the grip to move the line.

Grips do not appear on grading component lines that are constrained or locked by the grading criteria.

# Deleting Gradings

Use the Delete Gradings  command to delete a grading.

Surfaces based on deleted grading objects remain in the drawing.

**To delete a grading**

**1** Select the grading. Click Grading tab ➤ Modify panel ➤ Delete Grading ◺ .

**2** Select a point in the grading or site to delete the grading object.

### Quick Reference

Ribbon

Select the grading. Grading tab ➤ Modify panel ➤ Delete Grading ◺

Menu

Grading menu ➤ Edit Grading ➤ Delete Grading

Command Line

DeleteGradings

Supported Objects

■ Feature lines

■ Parcel lines

■ Survey figures

# Grading Properties

You can view and change grading group and grading properties.

# Grading Group Properties

Edits grading group properties.

These properties can be used to create the grading group and to control the surface properties of the grading group.

If you chose not to create a dynamic surface for the grading group when you created it, use the Grading Group Properties to create this surface. You can also specify the surface to use for volume calculations.

**To view and edit properties for a grading group**

1  In Toolspace, on the Prospector tab, expand the Site collection.

2  Expand the Grading Groups collection. Right-click the grading group you want to edit.

3  Click Properties to open the Grading Group Properties (page 1891) dialog box.

4  Change the properties on the Information tab, or click the Properties tab to review the settings, styles, criteria, and volumes for the grading group.

5  Click OK.

## Quick Reference

Ribbon

Select the grading. Click Grading tab ➤ Modify panel ➤ 🔲 Grading Group Properties

Toolspace Shortcut Menu

Prospector tab: Right-click Grading Group item ➤ Properties

Object Shortcut Menu

Right-click grading ➤ Grading Group Properties

Command Line

GradingGroupProperties

Dialog Box

Grading Group Properties (page 1891)

# Grading Object Properties

Edits the properties of a grading object.

Changes the cut and fill styles and can be used to move a grading to another grading group.

**To view and edit properties for a grading**

1  Select the grading. Click Grading tab ➤ Modify panel ➤ Grading Properties drop-down ➤ Grading Properties 🔲 .

2  In the Grading Creation Tools, click 🔲 .

3  In the drawing, click a grading to open the Grading Properties (page 1894) dialog box.

4  Change the styles or select a different grading group.

5  Click OK.

## Quick Reference

Ribbon

Select the grading. Click Grading tab ➤ Modify panel ➤ Grading Properties drop-down ➤ Grading Properties 🔲

Object Shortcut Menu

Right-click grading object ➤ Grading Properties

Command Line

GradingProperties

Dialog Box

Grading Properties (page 1894)

# Feature Line Properties

Edits the properties of a feature line.

Changes the name and style assigned to a feature line. You can also view feature line statistics and breakline data.

### To change feature line properties

**1** Right-click a feature line and click Feature Line Properties.

**2** In the Feature Line Properties dialog box, click the Information Tab (page 1910).

**3** Select the Name check box.

**4** Enter a name or click ![icon] to edit the name template in the Name Template Dialog Box (page 1806).

**5** Select the Style check box.

**6** Select a style.

**7** Click the Style Detail button to open the Style Detail dialog box. Preview the feature line style.

**8** Click the Statistics tab (page 1911) to view feature line statistics and breakline data.

### Quick Reference

Ribbon

Click Feature Line tab ➤ Modify panel ➤ Feature Line Properties drop-down ➤ Feature Line Properties ![icon]

Object Shortcut Menu

Right-click feature line ➤ Feature Line Properties

Command Line

FeatureLineProperties

Dialog Box

Feature Line Properties (page 1910)

# Feature Line Site Properties

Use the Feature Line Site Properties to view the combined statistics for all feature lines in the current site.

The Feature Line Site Properties dialog box Statistics tab displays the combined statistics for all feature lines in the current site. The Grouped Statistics tab groups the site statistics by style and layer. The statistics cannot be edited.

**To view current site feature line properties**

➤  In Toolspace, on the Prospector tab, right-click Feature Lines and click Properties.

### Quick Reference

Object Shortcut Menu

  In Toolspace, on the Prospector tab, right-click Feature Lines ➤ Properties

Dialog Box

  Statistics Tab (Feature Line Site Properties Dialog Box) (page 1914)

Dialog Box

  Grouped Statistics Tab (Feature Line Site Properties Dialog Box) (page 1914)

# Using Grading Utilities

You can use grading utilities to balance cut and fill volumes in a grading design, and to create a detached surface from a grading group.

# Displaying and Adjusting Surface Volumes

Use volume tools to display and adjust the elevation of a grading design.

You can change the elevation of either an entire grading group or a selection of grading objects. Use this command to optimize cut and fill requirements.

A dynamic surface must exist for the grading group and a Volume Base Surface must be specified in the Grading Group Properties to use most of the Grading Volume Tools.

---

**TIP** If you are designing a detention pond and want to size it by volume, it may help to create a temporary surface to use as the Volume Base Surface for comparison. To do this, draw a polyline around the pond rim, assign it the rim elevation, and add it as breakline data to a surface. When you use such a surface as the Volume Base Surface for the grading group, you can more easily see how much volume the pond can contain.

---

**To adjust cut and fill volumes**

1  Click Analyze tab ➤ Volumes And Materials panel ➤ Grading Volume Tools 🔷 .

2  In the Grading Volume Tools (page 1899), click 🔲 .

3  In the Select Grading Group (page 1900) dialog box, select a grading group or create a new one.

4  Optionally, click 🔲 to review and change properties for the grading group.

---

**NOTE** If you do not see the information displayed in the Cut, Fill, and Net volumes boxes, check the Grading Group Properties to ensure Automatic Surface Creation is selected and a Volume Base Surface is specified.

---

5  Select either Entire Group or Selection to identify the grading objects for which you want to raise or lower to make the volume adjustments. For Selection, click 🔲 and select objects in the drawing.

6  In the numeric field in the upper right of the Grading Volume Tools, enter the distance (typically feet or meters) by which the grading objects will be raised or lowered for each elevation adjustment.

**7** Do one of the following:

- To manually adjust the grading elevation, click  or  and note the changed values for cut, fill, and net volume.

- To automatically adjust the elevation to meet a net volume target, click . Enter the target volume in the Auto-Balance Volumes dialog box, then click OK. The elevation is adjusted several times, getting as close to the target as possible.

---

**NOTE** Automatic volume balancing is an iterative process, adjusting the gradings up or down in progressively smaller steps to achieve the target volume. Typically the resulting volume is not exactly what was entered as the target value. To prevent it from excessively adjusting and recomputing the volumes, it stops when the result is within a tolerance of 0.1% of the starting net volume. If the result is not close enough to your target volume, you can run the command again.

---

## Quick Reference

Ribbon

Analyze tab ➤ Volumes And Materials panel ➤ Grading Volume Tools 

Menu

Grading menu ➤ Grading Utilities ➤ Grading Volume Tools

Command Line

GradingVolumeTools

# Creating a Detached Surface from a Grading Group

You can create a detached or static surface from a grading group.

A detached surface does not update to reflect changes to the grading group.

If the Automatic Surface Creation setting is selected for the grading group, it maintains a dynamic surface that reflects changes.

If the grading group has a detached surface and a dynamic surface, it can be difficult to view and select one of the surfaces unless they are on different layers and using different styles.

**To create a detached surface**

**1** Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Grading .

**2** In the Select Grading Group (page 1900) dialog box, specify the site and the grading group.

**3** Click OK. The detached surface is created.

## Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Grading

Menu

    Grading menu ➤ Grading Utilities ➤ Create Detached Surface

Command Line

    CreateDetachedGradingSurf

# Grading Command Reference

The following table lists the grading-related commands and briefly describes their functionality.

Run these commands by entering them directly at the command line.

| Command | Description |
|---|---|
| AddFeatureLineLabels | Adds line and curve labels to feature lines (page 787) |
| AdjacentFeatureElevsByRef | Sets adjacent feature line elevations by reference (page 762) |
| ApplyFeatureLineNames | Applies names to feature lines (page 746) |
| ApplyFeatureLineStyles | Applies feature line styles (page 746) |
| BreakFeatures | Breaks a feature line into two feature lines (page 784) |
| ChangeGradingGroup | Changes the grading group of the selected grading object (page 742) |
| Convert3dPolys | Converts 3D polylines to 2D polylines (page 789) |
| ConvertPlines | Converts 2D polylines to 3D polylines (page 789) |
| CopyCreateGrading | Creates grading by copying the criteria and style from an existing grading (page 792) |
| CreateDetachedGradingSurf | Creates a static surface from a grading group. (page 803) |
| CreateFeatureLines | Creates a feature line by converting existing objects (page 743) |
| CreateFeatureLineFromAlign | Creates a feature line by converting alignments (page 745) |
| CreateGrading | Creates grading from a footprint (page 791) |
| CreateGradingGroup | Creates a new grading group (page 741) |
| CreateGradingInfill | Fills a hole in the site grading with a grading face (page 797) |
| CreateGradingTransition | Inserts a transition between two existing regions (page 794) |
| DeleteElevPoint | Deletes elevation points (page 767) |

| Command | Description |
|---|---|
| DeleteFeaturePI | Deletes points of intersection (page 771) |
| DeleteElevPoint | Deletes feature line elevation points (page 767) |
| DeleteGradings | Deletes a point in a grading or site (page 799) |
| DrawFeatureLine | Draws a feature line (page 748) |
| EditFeatureCurve | Edits feature line curves (page 773) |
| EditFeatureElevs | Edits feature line or lot line elevations, using the command line (page 753) |
| EditGrading | Edits grading criteria, using the command line (page 794) |
| EditPlineElevs | Edits elevations of 2D polylines (page 790) |
| FeatureElevsFromSurf | Creates feature line elevations from a surface (page 768) |
| FeatureGradeExtensionByRef | Extends a grading by reference (page 763) |
| FeatureLineProperties | Specifies feature line style (page 801) |
| FeatureLineRemLinks | Removes feature line dynamic links (page 747) |
| FilletFeature | Rounds the corners of feature lines (page 776) |
| FitCurveFeature | Converts tessellated curves to true arcs for better grading results (page 777) |
| GradingEditor | Edits grading criteria, using the Editor (page 793) |
| GradingElevEditor | Edits feature line elevations, using the Grading Elevation Editor (page 753) |
| GradingTools | Opens the Grading Creation Tools (page 791) |
| GradingVolumeTools | Opens the Grading Volume Toolbar (page 802) |
| InsertFeatureElevPoint | Inserts an elevation point (page 765) |
| InsertFeatureHighLowPoint | Inserts high or low elevation points into feature lines (page 767) |
| InsertFeaturePI | Inserts points of intersection (page 769) |
| JoinFeatures | Joins multiple feature lines to create a single object (page 771) |
| OffsetFeature | Offsets feature lines at an elevational difference (page 783) |

| Command | Description |
| --- | --- |
| QuickEditFeatureElevs | Edits the elevations of a feature line by snapping to editable points in the drawing (page 754) |
| RaiseLowerFeatures | Raises or lowers feature line elevations (page 764) |
| RaiseLowerFeaturesByRef | Raises or lowers feature line elevations by reference (page 761) |
| ReverseFeature | Reverses the direction of feature lines for stationing purposes (page 772) |
| SetFeatureGrade | Specifies the grade or slope between selected points on a feature line (page 758) |
| SetFeatureRefElev | Sets the feature line elevation in relation to an existing elevation in the drawing (page 759) |
| SmoothFeature | Smooths out jagged feature lines (page 779) |
| TrimFeatures | Trims a feature line (page 785) |
| WeedFeatures | Deletes vertices and elevation points from feature lines (page 781) |

# Parcels

# 22

AutoCAD Civil makes it easy to design subdivisions by providing a site topology that includes intelligent parcel objects, which maintains relationships between objects so your design changes are dynamically updated.

Parcels Workflow

Parcels Best Practices

## Understanding Parcels

Parcel objects in AutoCAD Civil are typically used to represent real estate parcels, such as lots in a subdivision.

Parcel objects can also represent other features with closed boundaries, such as bodies of water and soil regions.

## Parcel Components

A parcel boundary is a closed polygon. The points where line segments join are called nodes. There are two segment types: line and curve.
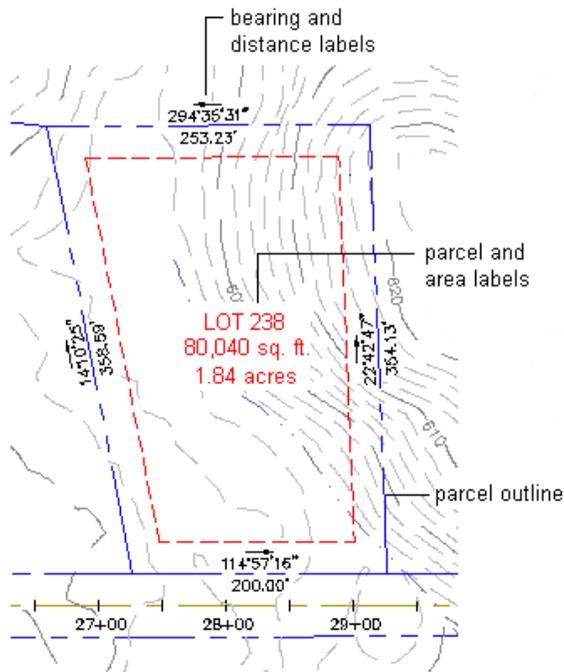
Each parcel has an area label, which refers to the parcel as a whole. You can click the area label to select the parcel. A parcel can also have segment labels, and associated tables, which display information about the parcel. Labels can contain user-defined property fields.

Each parcel maintains information about its segments and nodes, and the area they enclose. If you change one component, related components are updated.

# Parcel Collections

Parcels are contained in a parcels collection, which in turn is contained in a site.

A drawing can include any number of sites, but each site has only one parcels collection. No unique parcel can belong to more than one site. You can see these relationships in the Sites collection in Toolspace, on the Prospector tab:
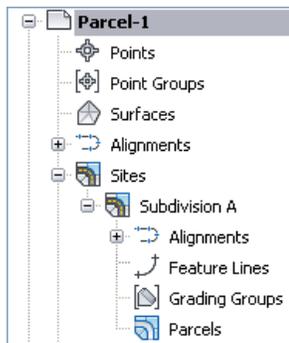


For more information about Parcels collections, see Parcels Collections (Prospector Tab) (page 811).

Parcels in a collection can be either unattached or touching, but they cannot overlap. If you attempt to overlap two parcels, the region of overlap defines a third parcel. Similarly, if an alignment crosses a parcel, it divides the parcel into two parcels. In the following illustration, before adding the alignment, the site was all one parcel:
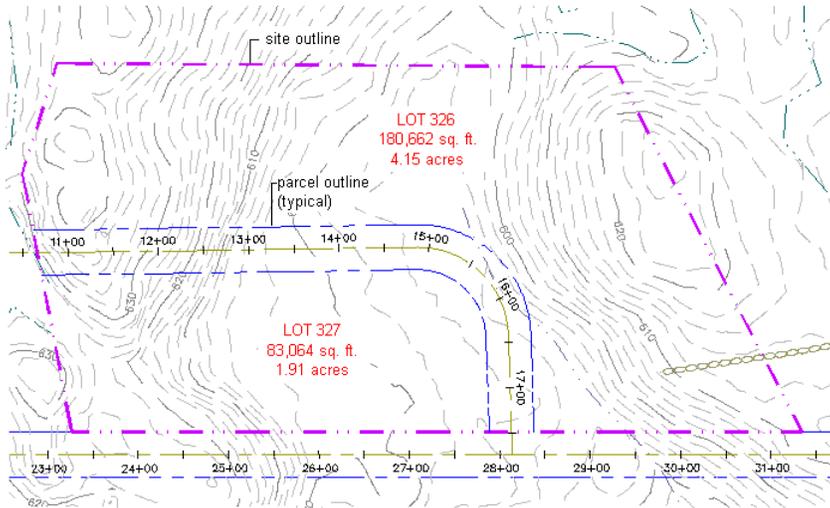
In situations that require overlapping parcels, such as overlaying a subdivision with soil regions, you can put the subdivision parcels in one site and the soil region parcels in another site. Multiple sites can overlay one another because their topologies are independent. Although everything within a site is topologically related, sites are not topologically related to each other.

**NOTE** To prevent an alignment from changing a parcel, the two objects should be in different sites. To prevent an alignment from interacting with any parcel objects, select <none> when prompted to select a site. See Understanding Alignment Objects (page 855).

# Site Parcels

Associated with every site is a site parcel. The site parcel boundary is the outermost extent of a site, containing all alignments, gradings, and parcels. Objects within a site parcel do not have to touch.
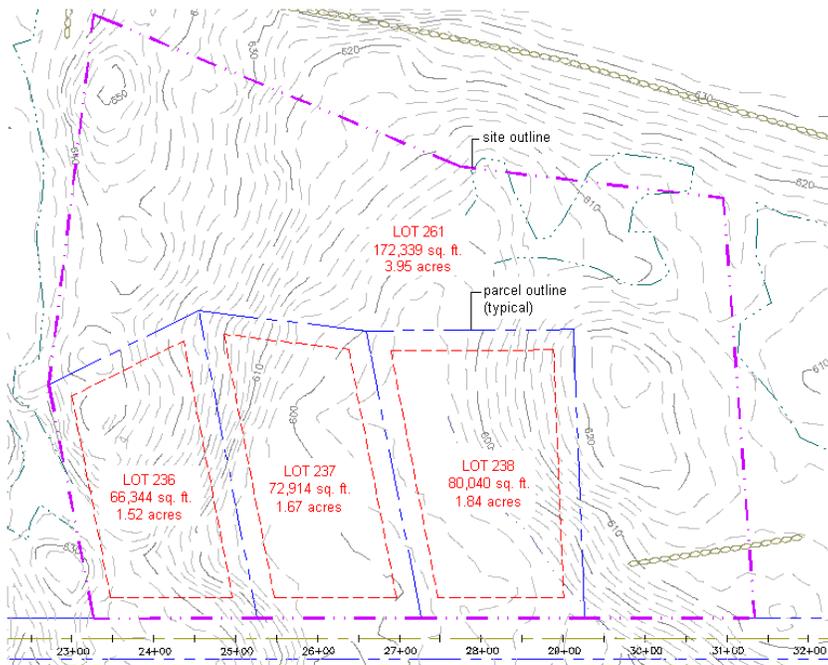
The site parcel has an area label, which you can hide or show. You can control the color and line type of the site parcel boundary, and you can control its draw order relative to the boundaries of the parcels that it encloses.

For the purposes of calculating the area and perimeter of a site, an alignments area is zero and its perimeter is twice the sum of its length. Adding an alignment does not affect a site parcels area, but can add to its perimeter. If you add an alignment to an existing site

- With *none* of the alignment overlaying the original site parcel, the site parcels perimeter increases by twice the length of the alignment.

- With *all* of the alignment overlaying the original site parcel, the site parcels perimeter does not increase.

- With *some* of the alignment overlaying and some of it projecting, the site parcels perimeter increases by twice the length of the part that projects.

In some circumstances, an unclosed sequence of parcel segments can define a valid parcel. With such a parcel, as with an alignment, its area is zero, and its perimeter is twice the sum of its segment lengths.

# Parcel Styles

Use styles to control the appearance of parcels, their labels, and their tables.

The Standard style is provided by default. Define additional styles as needed.

View the available parcel styles in the Parcels collection in Toolspace, on the Settings tab:
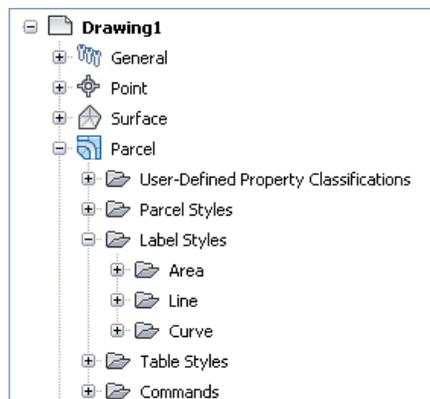


For more information about the Parcel collection, see Parcel Collection (Settings Tab) (page 812).

You can create, modify, copy, rename, or delete any style, including the Standard style, by right-clicking a style in the Parcel tree and then clicking an item on the shortcut menu. Note, however, that you cannot delete a style that is in use.

| For more information about ... | Follow this link ... |
|---|---|
| Setting up styles for a project | Setting up Parcels |
| Parcel styles | Parcel Styles (page 828) |
| Parcel label styles | Parcel Label Styles (page 833) |

# Parcel Settings

Use Parcel settings to specify default styles and other default properties for new parcels.

To set or change parcel settings, go to the Toolspace Settings tree and right-click Parcels ➤ Edit Feature Settings.

For more information about the Parcel collection, see Parcel Collection (Settings Tab) (page 812).

For more information about parcel settings, see Parcel Settings (page 824).

# Parcel Properties

Use Parcel properties to view or edit details of existing parcels.

To view or edit parcel properties, go to the Toolspace Prospector tab, right-click a parcel and click Properties. The following illustration shows two parcels: OPEN SPACE 100, and OPEN SPACE 101:
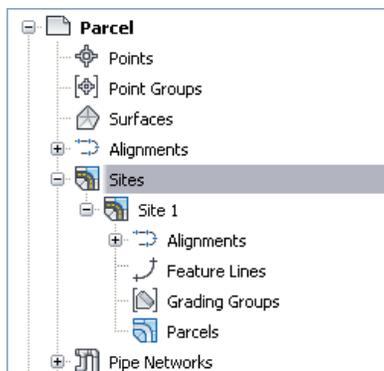


For more information parcel collections, see Parcels Collections (Prospector Tab) (page 811).

You can also right-click a selected parcel in a drawing, then click Parcel Properties. To select a parcel, click the text of its area label. If you have selected it correctly, the label text and its anchor point are highlighted. If the area label is not visible, see Hiding or Showing Parcel Labels (page 842).

For more information about parcel properties, see Parcel Properties (page 825).

# Parcels Collections (Prospector Tab)

Use the Parcels collections in the Prospector tree to access the parcels in a drawing.

Each parcel in a site is displayed as a named object under the Parcels collection of the site. Right-click a Parcels collection to do the following:

■  Edit properties of the collections site parcel (page 825).

■  Move or copy parcels to another site (page 726).

■  Zoom or pan to the collections site parcel in the drawing (page 819).

■  Generate a parcels report.

- Export parcels to LandXML format.

- Refresh the view of the Prospector tree.

Expand the Parcels collection to view the names of the parcels and to display a tabular list of the parcels in the Toolspace item view. For more information, see The Toolspace Item View (page 82).

## Parcel Collection (Settings Tab)

Use the Parcel collection in the Settings tree to manage parcel settings, styles, and command settings.

Right-click the Parcel collection to do the following:

- Edit parcel settings (page 824).

- Edit parcel label style defaults (page 832).

- Refresh the display of the Settings tree.

Expand the Parcel collection to display and edit the styles and command settings that are available for parcels.

| For more information about ... | Follow this link ... |
| --- | --- |
| Parcel Styles | Parcel Styles (page 828) |
| Parcel Label Styles | Parcel Label Styles (page 833) |
| Parcel Table Styles | Parcel Table Styles (page 843) |
| Parcel Commands | Parcels Command Reference (page 853)<br>Editing Parcel Command Settings (page 825) |

## Creating Parcels

Create parcels from drawing objects, by layout, or by subdividing an existing parcel.

When you create parcels from drawing objects, ensure that they have no drawing errors that would make them invalid topology sources. Use the drawing cleanup tools in Autodesk Map. When you create parcels by layout, you draw parcel segments directly.
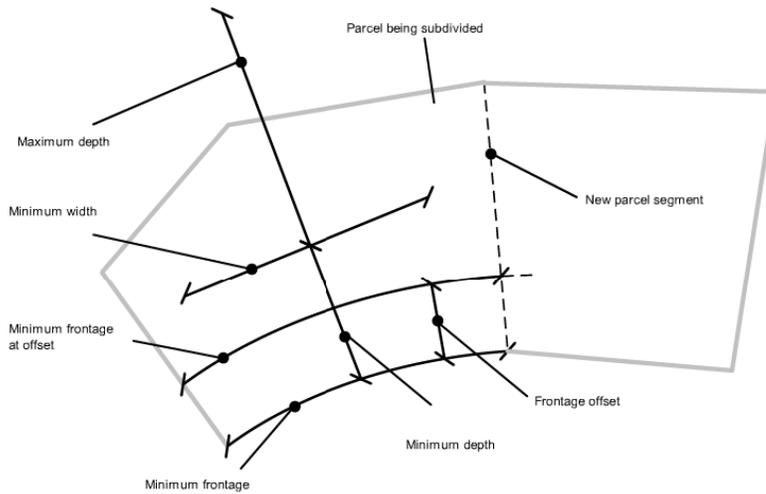
**You can define parcel sizing criteria when creating or editing parcels by subdividing existing parcels.**

# Parcel Design Considerations

Before creating parcels, identify the different types of parcels you need and decide how to display them so that different types are visually distinct.

You can design styles for parcels, labels, and tables. These styles control the appearance of the parcels and the information displayed in labels and tables. For more information, see Parcel Labels and Tables (page 831).

You may want to design more than one style for a given parcel type. For example, if your project drawings are consulted by several different users who want to see different information emphasized, use styles to provide customized views of the same drawing. Information that is visible in one set of styles can be hidden, de-emphasized, or emphasized in another.

After you design the required styles, you can save them in a drawing template. When you create a new drawing, you can base it on the template, which makes every style in the template available in the new drawing. The necessary styles for a project typically include parcel styles, area label styles, segment label styles, and table styles.

Extend the styles concept by using drawing layers to manage how features display. For example, a parcel style has a segment layer property, which assigns segments to a particular layer, and a segment color property. Specify the segment color property of a parcel style explicitly, or specify it By Layer, which lets you change the color of many segments at once by changing the color of their layer. As you can with styles, save layer definitions in a drawing template.

# Default Settings for New Parcels

The default properties for new parcels determine how new parcels are displayed. For more information, see Parcel Settings (page 824).

When you initiate a parcel creation session, you open the Create Parcels (page 2005) dialog box. This dialog box displays the default settings for new parcels. You can revise these settings before you create any parcels.

**To set how new parcels display**

1   Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Parcel From Objects ⬚ .

**2** If you clicked Create Parcel From Objects, click objects in the drawing to select them. When you finish, press Enter.

This step occurs only if you are creating from objects.

**3** In the Create Parcels (page 2005) dialog box, you can edit default settings before creating any parcels.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Parcel From Objects

Menu

Parcels menu ➤ Create Parcel From Objects

Command Line

CreateParcelFromObjects

Dialog Box

Create Parcels from Objects (page 2005)

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels menu ➤ Create Parcel By Layout

Command Line

CreateParcelByLayout

Dialog Box

Create Parcels by Layout (page 2005)

## Creating Parcels From Objects

Create parcels from AutoCAD objects by defining a selection set of objects in the drawing.

AutoCAD objects that you can convert to parcels include polygons, closed polylines, and other closed sequences of lines or arcs.

Select objects in the current drawing or from an xref. The objects you select must be free of drawing errors, such as gaps at intersection points.

---

**TIP** To remove drawing errors from source geometry, use the drawing cleanup tools in Autodesk Map before you import the objects.

---

**To create parcels from objects**

**1** Make sure that the AutoCAD objects that you are converting to parcels are free of drawing errors.

**2** Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Parcel From Objects .

**3** Click objects in the drawing to select them. When you finish, press Enter.

**NOTE** You can enter **xref** to select an object from an xref. The xref command supports multiple selections.

4 In the Create Parcels (page 2005) dialog box, you can edit default settings before creating any parcels. Click OK and the parcels are created.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Parcel From Objects 

Menu

Parcels menu ➤ Create Parcel From Objects

Command Line

CreateParcelFromObjects

Dialog Box

Create Parcels from Objects (page 2005)

# Creating Parcels by Layout

Use the Parcel Layout Tools toolbar to create parcels.

The toolbar provides two types of parcel creation tools:

■ Freehand tools draw parcels as a network of lines, curves, and polylines.

■ Precise sizing tools control the angles and directions of lot lines.

You can also set default values for the parcel sizing parameters and the Automatic Layout options.

**TIP** Object Snap can be very useful when creating parcels, especially with the freehand tools. Use Object Snap to attach lines exactly to existing endpoints and vertices.

**NOTE** Using the layout tools is the only activity possible while the Parcel Layout Tools toolbar is open.

**See also:**

■ Parcel Layout Tools (page 2009)

■ Editing Parcels by Layout (page 820)

**To create parcels by layout**

1 Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools  .

2 Click a parcel layout tool.

In the Create Parcels - Layout (page 2005) dialog box, you can edit default settings before creating any parcels.

3 Specify a site with which to associate the parcel or accept the default <None>. For more information, see Understanding Sites (page 719).

4 Specify a parcel style or accept the default style.

**5** Specify the Object Layer Setting.

**6** Specify the Parcel Label Set or accept the default. Click OK.

**7** Use the commands on the Parcel Layout Tools toolbar to create the parcel.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels menu ➤ Create Parcel By Layout

Command Line

CreateParcelByLayout

Dialog Box

Create Parcels by Layout (page 2005)

Dialog Box

Parcel Layout Tools (page 2009)

# Creating Parcels by Subdividing

Use the Parcel Layout Tools toolbar to create parcels by subdividing existing parcels.

When you add a new lot line to an existing parcel, you can specify the size of the new parcel and the method with which to move the lot line. You can insert a new free-form lot line or choose one of the following types that move across the parcel in a controlled manner:

■ Slide Line. Holds either a fixed angle to the frontage or an absolute direction.

■ Swing Line. Fixed at one end and swings across the parcel frontage

When creating or editing parcels with the Parcel Layout Tools, preview graphics display the proposed solution which results from the specified layout parameters used to define parcel components such as minimum parcel frontage, width, and depth. As you use the tools to determine the design of proposed new parcels, these graphics display to illustrate the proposed solution, and you are prompted to either accept or reject the solution. When you modify the layout parameters, the graphics update dynamically while the Parcel Layout Tools toolbar is open and the parameters are being edited. For more information, see Parcel Layout Tools (page 2009).
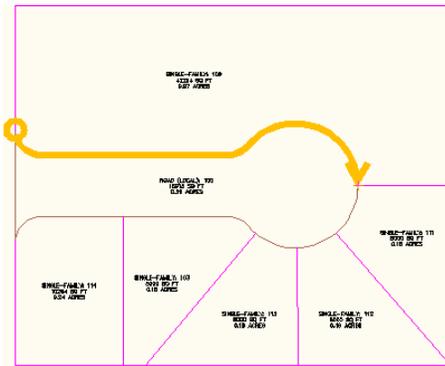
**Proposed frontage is highlighted.**
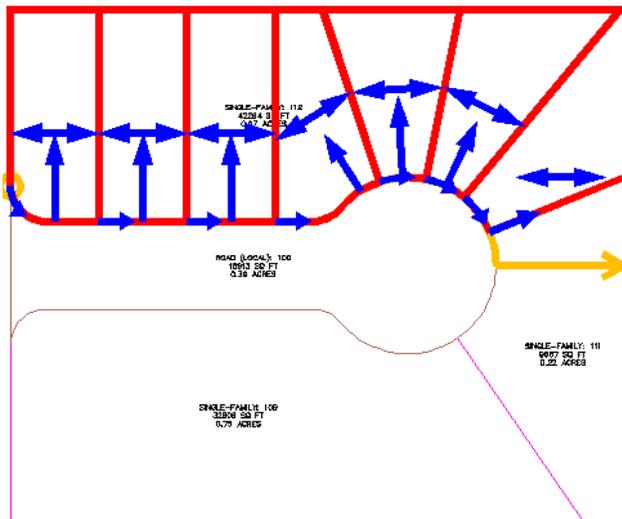


**Preview graphics display proposed layout solution based on parcel sizing parameter values.**

**See also:**

■ Parcel Layout Tools (page 2009)

**To create parcels by subdividing with the Slide Line - Create command**

**1** Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools ⊞ .

**2** On the Parcel Layout Tools toolbar, click ⏷ to expand the toolbar. Specify the Parcel Sizing and Automatic Layout parameters for the new parcel.

**3** Click ⊡ for the Slide Line - Create tool.

**4** In the Create Parcels – Layout (page 2005) dialog box, specify any changes to the default settings. Click OK.

**5** Click the Parcel Selection Area label to select the parcel or enter p and then click inside the parcel.

**6** Click the start point on the parcel frontage.

**7** Click an end point along the frontage. A preview graphic highlights the proposed frontage.

**8** Specify the angle, bearing or azimuth to place the new lot line. Preview graphics display if a solution can be obtained.

> **NOTE** When prompted for the angle, bearing or azimuth, you can press Enter to enter the default 90 degree angle.

**9** Accept the result to edit the parcel. If a solution based on the current parcel sizing parameter values is not found, you are prompted to adjust the parameters or reenter a new value for the method you are using.

**10** Optionally, continue creating lot lines by repeating Steps 5-9. When you finish, press Esc.

**To create parcels by subdividing with the Swing Line - Create command**

**1** Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools .

**2** On the Parcel Layout Tools (page 2009) toolbar, click to expand the toolbar. Specify the Parcel Sizing parameters and Automatic Mode options for the new parcel.

**3** Click for the Swing Line - Create tool.

**4** In the Create Parcels – Layout (page 2005) dialog box, specify any changes to the default settings. Click OK.

**5** Select the parcel to subdivide.

**6** Click the start point on the parcel frontage.

**7** Click an end point along the frontage. A preview graphic highlights the proposed frontage.

**8** Specify the swing point. Preview graphics display if a solution can be obtained.

**9** Accept the result to subdivide the parcel. If a solution based on the current parcel sizing parameter values is not found, you are prompted to adjust the parameters, select a new point, or exit the command.

**10** Optionally, continue creating lot lines by repeating Steps 4-10. When you finish, press Esc.

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels menu ➤ Edit Parcel ➤ Edit Parcel Segments

Dialog Box

Parcel Layout Tools (page 2009)

# Editing Parcels

Edit parcels to change their size, shape, or display properties.

Edit parcels in three main ways:

- Use drafting tools. Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools ▧ . and use the Parcel Layout Tools (page 2009).

- Change properties and labels. Click the area label to select the parcel, then right-click and select a menu option.

- Grip edit. Click lot lines and drag their nodes to new positions.

# Locating Parcels in a Drawing

Before you can select a parcel, locate it in the drawing. Locate it by selecting the parcel in the Parcels collection in Toolspace, on the Prospector tab, and then zooming or panning to it in the drawing.

**To locate a parcel**

1 In Toolspace, on the Prospector tab click Sites ➤ Parcels.

2 Right-click a Parcel. Click Zoom To or Pan To.

   To zoom or pan to the site parcel, right-click the Parcel. Click Zoom To or Pan To.

### Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Parcel ➤ Zoom To or Pan To

# Changing Parcel Appearance

Control parcel display by changing parcel styles or their label styles.

Within a given site, you can change the display order of the site parcel and the parcels that it contains.

| To change a parcels appearance by... | See this topic... |
| --- | --- |
| Editing parcel styles | Editing Parcel Styles (page 828) |
| Editing parcel label styles | Editing Parcel Label Styles (page 834) |
| Replacing parcel styles | Applying Parcel Styles (page 829) |
| Replacing parcel label styles | Applying Parcel Label Styles (page 837) |
| Changing display order | Changing Parcel Display Order (page 827) |

**See also:**

- Parcel Layout Tools (page 2009)

- Parcel Label Settings (page 832)

# Editing Parcels by Layout

Use the Parcel Layout Tools toolbar to edit parcels.

The toolbar provides tools for creating and editing parcels. For information about tools for parcel creation, see Creating Parcels by Layout (page 815).

The toolbar provides two types of parcel editing tools:

- **Precise sizing tools** control the angles and directions of lot lines.
- **Freehand tools** modify parcel geometry.

You can also modify settings for the Parcel Sizing parameters and Automatic Layout options.

Using layout tools is the only activity possible while the Parcel Layout Tools toolbar is open.

**See also:**

- Parcel Layout Tools (page 2009)

**To edit parcels by layout**

1 Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools .

2 Click a parcel segment to indicate the site you will be editing.

3 Click a parcel layout tool.

When you finish, press Enter or click another tool. After pressing Enter (which puts a tool away without selecting another), you can press Enter again to continue with the same tool.

4 When you finish editing parcels, press Esc to close the toolbar. (If one of the layout tools is still in use, press Esc twice.)

Using layout tools is the only activity possible while the Parcel Layout Tools toolbar is open.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools 

Menu

Parcels menu ➤ Edit Parcel ➤ Edit Parcel Segments

Command Line

EditParcel

Dialog Box

Parcel Layout Tools (page 2009)

# Editing Subdivided Parcels

Use the Slide Line and Swing Line edit commands to edit subdivided parcels.

You can use the Slide Line and Swing Line edit commands to edit attached lot lines. Attached lot lines are parcel segments that were created with the Parcel Layout Tools subdivision commands.

Using layout tools is the only activity possible while the Parcel Layout Tools toolbar is open.

**See also:**

■ Parcel Layout Tools (page 2009)

**To edit parcels by subdividing with Slide Line - Edit command**

1 Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools  .

2 Click  for the Slide Line - Edit tool.

3 In the Create Parcels - Layout dialog box, adjust default settings and press OK.

4 Select the attached lot line to adjust.

5 Click inside the parcel to be edited.

6 Click the start point on the parcel frontage.

7 Click an end point along the frontage. A preview graphic highlights the proposed frontage.

Specify the angle, bearing or azimuth to edit the lot line. Preview graphics display if a solution can be obtained.

**NOTE** When prompted for the angle, bearing or azimuth, you can press Enter to enter a 90 degree angle as this is the default angle.

8 Accept the result to edit the parcel. If a solution based on the current parcel sizing parameter values is not found, you are prompted to adjust the parameters or reenter a new value for the method you are using.

9 When you finish editing parcels, press Esc to close the toolbar. (If one of the layout tools is still in use, press Esc twice.)

**To edit parcels by subdividing with Swing Line - Edit command**

1 Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools  .

2 Click  for the Swing Line - Edit tool.

3 In the Create Parcels - Layout dialog box, adjust default settings and press OK.

4 Select the attached lot line to adjust.

5 Click inside the parcel to be edited.

6 Click the start point on the parcel frontage.

7 Click an end point along the frontage. A preview graphic highlights the proposed frontage.

**NOTE** OSNAP must be turned off when using the Swing Line commands.

8 Specify the swing point. Preview graphics display if a solution can be obtained.

9 Accept the result to edit the parcel. If a solution based on the current parcel sizing parameters values is not found, you are prompted to adjust the parameters or reenter a new value for the method you are using.

10 When you finish editing parcels, press Esc to close the toolbar. (If one of the layout tools is still in use, press Esc twice.)

**Quick Reference**

Ribbon

Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels menu ➤ Edit Parcel ➤ Edit Parcel Segments

Command Line

EditParcel

Dialog Box

Parcel Layout Tools (page 2009)

# Creating a Right of Way

Use the Create ROW command to create a right of way along an alignment.

Parcels and adjacent alignment must be in the same site. Parcel boundaries are offset a specified distance from the alignment.

Optionally, specify a radius for fillets at intersections with other objects. The following illustration shows a right of way created between parcels 103 and 104 with fillets at intersections with parcel boundaries.
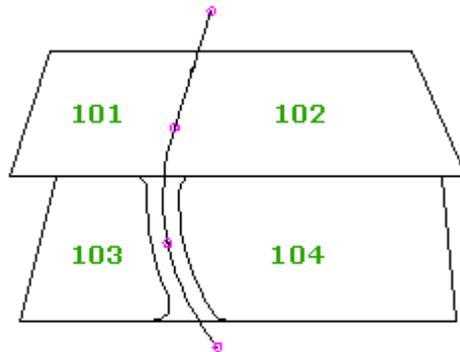


The right of way is like a narrow parcel, but it is not dynamically linked to the alignment. If you move or edit the alignment you must create the right of way again.

When you run the Create ROW command, you are prompted to select one or more parcels. If an alignment is found in the site and along an edge of the selected parcels, a right of way is created in accordance with the supplied parameters.

**To create a right of way**

1   Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Right Of Way .

2   Select one or more parcels in the drawing and press Enter.

3   In the Create Right of Way (page 2020) dialog box, specify the offset distance from the alignment to the parcel boundaries.

4   If you want corner fillets at parcel boundaries along the right of way, select this option. Specify a radius.

5   If you want corner fillets at intersections with other alignments and rights of way, select this option. Specify a radius.

**6** Click OK.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Right Of Way

Menu

Parcels menu ➤ Create ROW

Command Line

CreateParcelROW

Dialog Box

Create Right of Way dialog box (page 2020)

# Deleting Parcels

Delete a parcel only if it has one or more segments that are not shared with another parcel.

**NOTE** If you delete a segment that is shared by two parcels, you merge the two parcels. See Merging Parcels (page 824).

**To delete a parcel**

**1** Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools .

**2** Click a parcel segment to indicate the site you will be editing.

**3** On the Parcel Layout Tools toolbar, click . Click a parcel segment that is not shared by another parcel.

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels ➤ Edit Parcel ➤ Edit Parcel Segments

Dialog Box

Parcel Layout Tools (page 2009)

Parcel Layout Tools Toolbar

Command Line

ERASE

# Merging Parcels

To merge two parcels, delete a shared segment.

When you delete a shared segment, you delete the shared boundary. The two parcels become one.

If the shared boundary is composed of more than one segment, all the shared segments are deleted, not just the one you selected.

---

**NOTE** If you delete a segment that is not shared, you delete the parcel that it belongs to. See Deleting Parcels (page 823).

---

1 Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools  .

2 Click a parcel segment to indicate the site you will be editing.

3 In the Parcel Layout Tools toolbar, click  . Click a parcel segment that is shared by two parcels that you want to merge.

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools 

Menu

Parcels ➤ Edit Parcel ➤ Edit Parcel Segments

Dialog Box

Parcel Layout Tools (page 2009)

Parcel Layout Tools Toolbar



# Editing Parcel Geometry and Parcel Elevations

Use the feature line editing commands to edit parcel geometry and parcel elevations.

You can access these commands by selecting a parcel and then selecting either the Edit Geometry or Edit Elevations panels to access the commands. For more information, see Editing Feature Lines (page 751).

# Parcel Settings

Parcel settings include settings for parcels, parcel commands, and parcel labels.

This section includes information about parcel settings and parcel command settings. For information about Parcel label settings, see Parcel Label Settings (page 832).

# Editing Parcel Settings

Parcel settings specify the default properties for new parcels. Edit parcel settings in the Parcel collection, in Toolspace on the Settings tab.

**To edit parcel settings**

1   In Toolspace, on the Settings tab, right-click Parcel ➤ Edit Feature Settings.

2   In the Edit Feature Settings dialog box, edit parcel settings.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Parcel ➤ Edit Feature Settings

Dialog Box

Parcel Settings (page 1999)

# Editing Parcel Command Settings

Parcel command settings specify the default options for parcels commands. Edit parcel command settings under Commands collection in Toolspace on the Settings tab.

**To edit parcel command settings**

1   In Toolspace, on the Settings tab, expand Parcel ➤ Commands.

2   Right-click a command. Click Edit Command Settings.

3   In the Edit Feature Settings - Parcel (page 1999) dialog box, edit command settings.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Parcel Commands ➤ Edit Command Settings

Dialog Box

Parcel Settings (page 1999)

# Parcel Properties

Properties define each parcel in a drawing.

Properties control everything about a parcel except its segment label styles. A site parcel has an additional property that specifies the display order of parcels within the site, including the site parcel itself.

For information about replacing parcel segment label styles, see Applying Parcel Label Styles (page 837).

# Editing Parcel Properties

Edit parcel properties to work with styles and other basic parcel attributes.

You can edit the properties of a single parcel, or a group of parcels. In each case, you can change the parcel style, area label style, and user defined properties such as parcel number and address.

Some typical editing tasks are as follows:

■   Replace a parcel's style, which governs the appearance of the parcel itself.

- Replace a parcel's area label style, which governs the appearance of its area label and the information displayed in the label.

- View information about the parcel segments.

- Specify the parcel's Point Of Beginning, which is the parcel boundary's initial node (start point of its initial segment).

- Renumber parcels to reset their starting and increment values and/or rename a parcel based on the parcel name template.

- Change parcel display order to set the properties of shared segments (such as color) when the two parcels sharing it have different properties.

Edit parcel properties by right-clicking a parcel area label in the drawing, or by right-clicking a parcel in Toolspace on the Prospector tab (or a Parcels collection for a site parcel).

**To edit properties of one parcel**

1   Click the text of the parcels area label.

    If the parcels area label is not visible, see Hiding or Showing Parcel Labels (page 842).

    If you have selected the label correctly, the label text and its anchor point are highlighted. Selecting the area label also selects the parcel.

2   Right-click and click Parcel Properties.

3   In the Parcel Properties (page 2015) dialog box, edit parcel properties.

**To edit properties of multiple parcels**

1   Click Parcel tab ➤ Modify panel ➤ Parcel Properties drop-down ➤ Parcel Properties 📑 .

2   The current site is displayed and the following prompt is displayed:

    `Start point or [Polyline/All/Site]:`

3   Do one of the following:

    - Select a start and end point to draw a path through the desired parcels.

    - Select **Polyline**, to select a polyline that crosses through the desired parcels. The polyline prompt will repeat until you hit **Enter** to end the command.

    - Select **All**, to select all parcels and open the Edit Parcel Properties (page 2008) dialog box.

    - Select **Site**, to open a Select Object (Site) dialog box with which to select another available site. After selecting a site, start back at Step 2.

4   In the Edit Parcel Properties dialog box, edit the parcel properties.

## Quick Reference

Ribbon

Select the parcel. Click Parcel tab ➤ Modify panel ➤ Parcel Properties drop-down ➤ Parcel Properties 📑

Menu

Parcels menu ➤ Edit Parcel ➤ Edit Parcel Properties

Commands

**EditParcelProperties** for one parcel
**ParcelPropertiesEdit** for multiple parcels

Dialog Boxes

Edit Parcel Properties Dialog Box (page 2008)
Parcel Properties Dialog Box (page 2015)

# Changing Parcel Display Order

Control parcel style display order, which is a property of the site parcel.

Display order affects the display of all parcel segments that are shared by two parcels of different styles. The style that is higher in the display order controls the actual display.

Edit the parcel style display order as a property of the site parcel, using the Composition Tab (page 2017) of the Site Parcel Properties dialog box. Access this dialog box by right-clicking either the site parcels area label or the Parcels collection in the Toolspace on the Prospector tab.

**To change parcel display order**

1   Click the site parcels area label in the drawing.

    If the parcels area label is not visible, see Hiding or Showing Parcel Labels (page 842).

    If you have selected the label correctly, the label text and its anchor point alone are highlighted. Selecting the area label also selects the parcel.

2   Right-click and click Parcel Properties.

3   In the Site Parcel Properties dialog box, click the Composition tab.

4   On the Composition tab, under Parcel Style Display Order, change the list positions of styles listed there.

    To change a styles list position, select it. Click the Up or Down Arrow. You can also drag a style to a new position. When you click Apply or OK, the parcels are redrawn, starting with the styles at the bottom of the list and working up to the top.

## Quick Reference

Ribbon

Select the parcel. Click Parcel tab ➤ Modify panel ➤ Parcel Properties drop-down ➤ Parcel Properties 

Object Shortcut Menu

Site parcel object  ➤  Parcel Properties

Toolspace Shortcut Menu

Prospector tab: Parcels  ➤  Properties

Dialog Box

Site Parcel Properties (page 2017)

# Parcel Styles

Parcel styles govern how parcels are displayed in a drawing. Create, edit, or delete parcel styles in the Parcel collection in Toolspace, on the Settings tab.

You can create an entirely new parcel style, or base it on an existing style.

The default Standard parcel style is always available. If you base a new drawing on a drawing template, the styles defined in the template are available also.

## Creating Parcel Styles

Create parcel styles in the Parcel collection in Toolspace, on the Settings tab.

**To create a parcel style**

1  In Toolspace, on the Settings tab, expand the Parcel collection.

2  Right-click Parcel Styles and click New.

3  In the Parcel Styles dialog box, name the new style and define the style settings.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Parcel Styles ➤ New

Dialog Box

Parcel Styles (page 2004)

## Editing Parcel Styles

Edit parcel styles on the Toolspace Settings tab, in the Parcel collection.

Edit the style name, parcel pattern fill, and displayed elements.

You can also edit the hatch pattern for the parcel pattern fill.

**To edit a parcel style**

1  In Toolspace, on the Settings tab, expand Parcel ➤ Parcel Styles.

2  Right-click the style that you want to edit and click Edit.

3  In the Parcel Styles dialog box, modify style settings.

### Quick Reference

Ribbon

Select the parcel. Click Parcel tab ➤ Modify panel ➤ Parcel Properties drop-down ➤ Edit Parcel Style

Toolspace Shortcut Menu

Settings tab: Right-click a parcel style ➤ Edit

## Creating Parcel Styles Based on Other Styles

Create a new parcel style by copying and modifying an existing style in the Parcel collection in Toolspace on the Settings tab.

**To create a parcel style based on another style**

1 In Toolspace, on the Settings tab, expand Parcel ➤ Parcel Styles.

2 Right-click the style that the new style will be based on and click Copy.

3 In the Parcel Styles dialog box, name the new style.
The settings for the new style will be the same as the settings of the style that you copied.

4 Modify style settings.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click a parcel style ➤ Copy

Dialog Box

## Applying Parcel Styles

Apply a different style to a parcel any time that you want to change its appearance or indicate a change in the type of parcel.

The two components of a parcel style that display are the parcel segment line and the area fill. Area fill is a hatch pattern that you can apply to the entire parcel area, or use to mark a border around the inside of the parcel boundary. You can set a fill distance to define the width of the filled border.

For information about applying parcel styles during the creation process, see Default Settings for New Parcels (page 813).

**To apply a parcel style to an existing parcel**

1 In the drawing, click a parcel area label to select the parcel.
If the label is not visible, see Hiding or Showing Parcel Labels (page 842).
If you have selected the label correctly, the label text and its anchor point are highlighted.

2 Right-click, then click Parcel Properties.

3 In the Parcel Properties dialog box, click the Information tab.

4 On the Information tab, under Object Style, specify the parcel style you want to apply.

### Quick Reference

Ribbon

Select the parcel. Click Parcel tab ➤ Modify panel ➤ Parcel Properties drop-down ➤ Parcel Properties 

Object Shortcut Menu

Parcel object ➤ Parcel Properties

Toolspace Shortcut Menu

Prospector tab: Parcel ➤ Properties

Dialog Box

Parcel Properties (page 2015)

## Deleting Parcel Styles

Delete parcel styles in the Parcel collection in Toolspace on the Settings tab.

**NOTE** The Standard style can only be deleted if it is not being referenced by an existing parcel.

**To delete a parcel style**

1  In Toolspace, on the Settings tab, expand Parcel ➤ Parcel Styles.

2  Right-click the style that you want to delete, then click Delete.
   Delete is unavailable for any style that is used in the drawing.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: a parcel style ➤ Delete

# Exporting a Parcel Inverse or Mapcheck Report

You can export an inverse or mapcheck report for a single parcel, for a selection of parcels, or for all site parcels at the same time.

You can access the Export Parcel Analysis (page 2019) dialog box from three locations within the Prospector tree: the main Parcels node, from individual parcels, or by selecting multiple parcels from the Parcel list view.

To export all of the parcels in a site, you can access the Export Analysis command from the main Parcels node.

To export an individual parcel report, right-click a parcel under the Parcels node.

To export a selection of parcels, use Ctrl-click to select multiple parcels from the Parcel list view that appears below the Prospector tree when the Parcels node is selected.

To run a Mapcheck Analysis on parcel labels, select the parcel labels and click Analyze tab ➤ Ground Data panel ➤ Survey Drop-down ➤ Mapcheck. For more information, see Performing a Mapcheck Analysis (page 1608).

**To export an inverse or mapcheck report**

1   In Toolspace, on the Prospector tab, do one of the following:

■ Expand a site and right-click the Parcels node.

■ Expand a site and expand the main Parcels node, and then right-click an individual parcel.

■ Use Ctrl-click to select multiple parcels in the Parcel list view and right-click.

2   Select Export Analysis.

3   In the Export Parcel Analysis (page 2019) dialog box, specify the path to where the file will be exported and saved. Enter the path or click 🗁 to browse to a location.

4   Select the analysis type.

5   Select or clear the Enable Mapcheck Across Chord check box.

6   Select the segment order.

7   Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click individual parcel, main Parcels node, or a selected group of parcels ➤ Export Analysis

Dialog Box

Export Parcel Analysis Dialog Box (page 2019)

# Parcel Labels and Tables

Both parcel labels and tables display parcel information.

Labels display information within the parcel or adjacent to the component they describe. Tables are constructed with one row for each parcel or component.

Parcels always have area selection labels. Segment labels are optional. You can label parcel segments when you create them or at any time afterward.

Styles control the appearance of labels and tables and the information they contain.

### Using Labels and Tables

Display parcel information in labels only, or in both labels and tables. Parcel tables are optional, but useful for organizing data when you work with a large number of parcels.

Parcel line, curve, and area labels that are added to already created parcels are now individual objects, not sub-objects of the parcel. These parcel labels can be controlled with the AutoCAD Properties palette. The first area label that is automatically created when the parcel is created is a sub-entity of the parcel and is used to select the parcel. This parcel selection label cannot be deleted.

If you want to use parcel tables, all parcels and segments to be included in a table require a label that displays a unique table tag. This tag serves as a unique identifier for the object in the drawing and in the table. The label can include additional information, but the tag is essential.

General line and curve label styles can be applied to parcel segment labels. As a result, when creating parcel tables, you can select general line/curve label types and/or parcel segment labels. You can select any parcel

segment labels that use line/curve styles. Or you can use any line or curve label that is applied to a line, curve, feature line, or polyline segment.

For more information about parcel labels, tables, and table tags, see Parcel Label Styles (page 833), Adding Parcel Tables (page 845), and Parcel Numbering and Naming (page 849).

### More About Labels and Tables

The topics in this section provide information about labels and tables in a parcels context. For information about labels in general, see Labels and Tags (page 1463). For information about tables, see Tables (page 1555). For information about modifying labels, see Modifying Labels in a Drawing (page 1533).

# Parcel Label Settings

Parcel label settings provide default settings for parcel label styles.

Global settings apply to parcel label styles of every type; label-type settings apply only to parcel label styles of a given type, such as area label styles.

Individual label styles inherit from label-type settings, which in turn inherit from global settings. You can override inherited settings at lower levels. Also, you can block overrides by locking settings at higher levels.

If a parcel label setting is changed at a higher level, the change is propagated downward except where it is overridden. The change affects all styles that inherit it, and all labels in the drawing that use those styles.

## Editing Global Settings for Parcel Label Styles

Global settings affect parcel label styles of every type. Edit these settings in the Parcel collection in Toolspace, on the Settings tab.

### To edit global settings for parcel label styles

1  In Toolspace, on the Settings tab, right-click Parcel and click Edit Label Style Defaults.

2  In the Edit Label Style Defaults dialog box, edit label style settings.

### Quick Reference

Toolspace Shortcut Menu

  Settings tab: Parcel ➤ Edit Label Style Defaults

Dialog Box

  Edit Label Style Defaults (page 1941)

## Editing Label-Type Settings for Parcel Label Styles

Label settings affect only parcel label styles of a given type: area, line, or curve.

Change label settings in the Label Styles collection, in Toolspace on the Settings tab. The parcel label types Area, Line, and Curve, are located under the Label Styles collection. These settings inherit values and overrides from global settings.

### To edit settings for parcel label styles

1  In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

**2** Right-click either an Area, Line, or Curve label style. Click Edit Label Style Defaults.

**3** In the Edit Label Style Defaults dialog box, edit parcel label style settings.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Label Styles ➤ Area, Line, or Curve. Right-click ➤ Edit Label Style Defaults

Dialog Box

Edit Label Style Defaults (page 1942)

# Parcel Label Styles

Parcel label styles control the appearance of parcel labels and the information they contain.

Parcel label styles include styles for area labels and segment labels (line labels and curve labels). Specify which label styles to use when you create parcels, and when adding labels. Line and curve general label styles can also be applied to parcel segment labels.

You can create, edit, or delete parcel label styles in the Label Styles collection in Toolspace, on the Settings tab. Create an entirely new parcel label style, or create one based on an existing style.

### Specifying Label Contents

Parcel label styles specify label contents that are composed of several information components. These components are identified in the Component list on the Layout tab of the Label Style Composer dialog box (page 1942). You can add or delete information components as needed, except with segment label styles, which have a permanent Table Tag component. The Table Tag component is essential for correlating segments in the drawing with those in a segment table.

When a given information component is selected in a styles Component list, its settings are displayed below it. Edit the Content setting to specify the components information. Edit other settings to specify how the component is displayed, including its relative position. You can preview a label in the Preview pane of the Layout tab.

Labels for line and curve segments include a property to span an outside segment. Set this property to True to apply a single label to any segment that runs along the outside edge of multiple parcels. Set this property to False to label the segment for each parcel separately. Set this property on the Layout tab of the Label Style Composer dialog box.

### Segment Labels and Tags

Segment label styles (unlike area label styles) include a Display Mode value, either Label or Tag. Labels are typically used to display engineering data adjacent to each segment. They can include multiple lines of text. Tags are generally much smaller. They display a reference number and perhaps one other element such as a direction arrow. The tag numbers appear in the first column of a table that contains the engineering data. By using tags, you can eliminate parcel segment data from the drawing and organize the data in a separate table. Within a drawing, you can use labels on some segments and tags on others.

Each information component in a segment label styles label template specifies a Used In value: Label Mode, Tag Mode, or Label And Tag Modes. These settings can help you assign data to either labels, tags, or both.

**Labels, Table Tags, and Tables**

To include a segment in a table the segment must have a style with the value set to Tag for the Display Mode property. The Table Tag component must have its Used In property set to either Tag Mode or Label And Tag Modes.

Area label styles do not have a Display Mode property. You can use any piece of information as the table tag. Typically the parcel number is used, but it can be another information component. The table tag, whatever it is, must also be included in the first cell of the corresponding table row. This information is specified in the tables style. For more information, see Adding Parcel Tables (page 845).

## Creating Parcel Label Styles

Create parcel label styles in the Label Styles collection in Toolspace, on the Settings tab.

**To create a parcel label style**

1   In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

2   Right-click a label type: Area, Line, or Curve and click New.

3   In the Label Style Composer dialog box, specify a style name and the style settings.

### Quick Reference

Ribbon

Select the parcel. Click Labels tab ➤ Modify panel ➤ Label Properties drop-down ➤ Edit Label Style 

Toolspace Shortcut Menu

Settings tab: Right-click a label type: Area, Line, or Curve ➤ New

Dialog Box

Label Style Composer (page 1942)

## Editing Parcel Label Styles

Edit parcel label styles in the Label Styles collection in Toolspace, on the Settings tab or by selecting the Edit Label Style option in the drawing.

**To edit a parcel label style**

1   In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

2   Expand a label type: Area, Line, or Curve.

3   Right-click the label style and click Edit.

4   In the Label Style Composer dialog box, modify style settings.

OR

1   Select the segment label to edit.

2   Right-click and select Edit Label Style.

3   In the Select Style dialog box, specify a label style, create a new style, or copy or edit an existing style.

## Quick Reference

## Editing Parcel Area Label Styles

Edit parcel area label styles in the Label Styles collection in Toolspace, on the Settings tab or by selecting the right-click Edit Area Selection Label Style option in the drawing.

You can select a different label style, edit a style, or create a new one.

**To edit a parcel area selection label style**

1  In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

2  Expand a label type: Area.

3  Right-click the label style and click Edit.

4  In the Label Style Composer dialog box, modify style settings.

OR

1  Select the area label to edit.

2  Right-click and select Edit Area Selection Label Style.

3  In the Select Style dialog box, specify a label style, create a new style, or copy or edit an existing style.

**NOTE** The Parcel Area Selection label cannot be deleted as this is an embedded object and is used to select the parcel.

## Quick Reference

**See also:**

■ Composition Tab (Parcel Properties Dialog Box) (page 2016)

## Displaying Parcel Area Label Properties with the List Command

Use the AutoCAD List command to display parcel area label property information.

**To display parcel area properties in the AutoCAD Text Window**

1   In the drawing, CTRL-click to select an area label.

2   In the command line, enter list.

3   The AutoCAD Text Window dialog box displays parcel properties, including diameter, area and segment information.

**To display parcel area label properties in the AutoCAD Text Window**

1   In the drawing, select an area label.

2   In the command line, enter list.

3   The AutoCAD Text Window dialog box displays the label type, and layer. For example, **AECC_PARCEL_FACE_LABEL Layer: "0"**.

### Quick Reference

Command Reference

   List

Dialog Box

   AutoCAD Text Window

## Creating Parcel Label Styles Based on Other Styles

Create a parcel label style based on another style in the Label Styles collection in Toolspace, on the Settings tab.

**To create a parcel label style based on another style**

1   In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

2   Expand a label type: Area, Line, or Curve.

3   Right-click the label style that the new style will be based on and click Copy.

4   In the Label Style Composer dialog box, specify a name for the new style.

5   Modify style settings.

### Quick Reference

Toolspace Shortcut Menu

   Settings tab: Right-click <label style> ➤ Copy

Dialog Box

   Label Style Composer (page 1942)

# Deleting Parcel Label Styles

Delete parcel label styles in the Label Styles collection in Toolspace, on the Settings tab.

**To delete a parcel label style**

1 In Toolspace, on the Settings tab, expand Parcel ➤ Label Styles.

2 Expand a label type: Area, Line, or Curve.

3 Right-click the style that you want to delete and click Delete.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <label style> ➤ Delete

# Applying Parcel Label Styles

A parcel area label style, which you can change at any time, is applied to parcels when they are created.

For information about applying parcel label styles during the creation process, see Default Settings for New Parcels (page 813).

Parcel area selection labels are always created and cannot be deleted. Parcel segment labels are not always applied in the creation process. Parcel segment labels are individual objects and can be placed on a layer separate from the segment being annotated. For more information on how to apply segment labels to existing parcels, see Adding Multiple Parcel Segment Labels (page 839).

**To replace a parcel label style**

1 Click a parcel label in the drawing.

If the label is not visible, see Hiding or Showing Parcel Labels (page 842).

If you have selected the label correctly, the label text and its anchor point are highlighted.

2 Right-click and click Label Properties or Properties.

3 In the AutoCAD Properties palette, replace the label style.

---

**NOTE** Use the Properties palette Label Style drop down menu to access a list of available parcel styles including the general line and curve label styles which can be used to annotate parcels.

---

## Quick Reference

Ribbon

Select the parcel label. Click Labels tab ➤ Modify panel ➤ Label Properties drop-down ➤ Edit Label Style

Object Shortcut Menu

Parcel object ➤ Label Properties or Properties

Dialog Box

AutoCAD Properties palette

# Adding or Deleting Parcel Labels

Add or delete parcel segment labels, but not area labels.

For information on how to change parcel area labels, see Applying Parcel Label Styles (page 837).

On the Toolspace Settings tab, in the General Label Style collection, set up Line and Curve label styles (page 1479) to use for the parcels.

On the Toolspace Settings tab, in the General Label Style collection, specify the default line label style (page 1476).

Use the Single Segment or Multiple Segment command to quickly label a parcel using the default label style.

Use the Add Parcel Labels option to access the Add Labels dialog box (page 1970) if you need to create or modify a style before labeling a feature line.

In the Add Labels dialog box you can specify the label types and styles to apply. You can also configure table tags and reference text object prompt methods.

You can add segment labels in the parcel creation process, or you can add them later. For information about adding segment labels during the creation process, see Default Settings for New Parcels (page 813).

## Adding Individual Parcel Segment Labels

You can label individual parcel segments with General Line and Curve label styles or Parcel Line and Curve label styles.

Parcel segment labels are independent, selectable objects. Segment labels can be placed on a layer separate from the corresponding segment. You can add segment labels automatically when you create a parcel. See Default Settings for New Parcels (page 813).

**To label parcel line segments**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Single Segment 🏷 .

2 Select the parcel line or segment to label.

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels 🏷 .

2 In the Add Labels dialog box, for Label Type, select Single Segment.

3 In the Add Labels dialog box, set Line Label Style and Curve Label Style and click Add.

> **NOTE** You can label parcel segments with either General Line and Curve label styles or with Parcel Line and Curve label styles.

4 Click a parcel segment in the drawing. The label is placed at the selected location. Click additional segments if you want.

While you select segments, the dialog box remains open. You can return to it to change styles or settings for the next sequence of clicks. After changing a label style, click Add. If you do not click Add before continuing to click segments, the style change will be ignored.

5 When you finish adding segment labels, click Close.

**Quick Reference**

## Adding Multiple Parcel Segment Labels

Add labels to multiple parcel segments in one operation.

The segments are labeled in either a clockwise or counterclockwise direction, regardless of the direction of individual segments. Labeling direction is set in the Default Options for the AddParcelSegmentLabels command.

You can add segment labels automatically when you create a parcel. See Default Settings for New Parcels (page 813).

**To label multiple parcel segments**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Multiple Segments 🏷 .

2   Select the feature lines or segments to label.

    When parcel face is selected, a direction prompt displays. If a different supported object is selected, the direction prompt does not display.

OR

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels 🏷 .

2   In the Add Labels dialog box, for Label Type, select Multiple Segment to add labels to all segments in a parcel.

3   In the Add Labels dialog box, set Line Label Style and Curve Label Style and click Add.

    **NOTE** You can label parcel segments with either General Line and Curve label styles or with Parcel Line and Curve label styles.

4   Click a parcel area label in the drawing. Labels are added to every segment of the parcel.

    While you select segments, the dialog box remains open. You can return to it to change styles or settings for the next sequence of clicks. After changing a label style, click Add. If you do not click Add before continuing to click segments, the style change will be ignored.

5   When you finish adding segment labels, click Close.

**Quick Reference**

## Replacing Multiple Segment Labels

Use the Replace Multiple Segment command to easily replace multiple parcel segment labels.

**To replace parcel segment labels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels .

2   In the Add Labels dialog box, for Label Type, select Replace Multiple Segment.

3   You are prompted to select a parcel to be labelled. It will replace the labels with new ones using the current Multiple Segment label style. If the Multiple Segment label style has already been added to the selected parcel a message will be displayed and the labels will not be replaced.

   While you select parcels, the dialog box remains open. You can return to it to change styles or settings for the next sequence of commands. After changing a label style, click Add. If you do not click Add before continuing to click segments, the style change will be ignored.

4   When you finish replacing multiple segment labels, click Close.

### Quick Reference

Ribbon

   Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels 

Menu

   Parcels menu ➤ Add Parcel Labels ➤ Add Parcel Labels

Command Line

   AddLabels

## Adding Parcel Area Labels

Add multiple area labels to a single parcel.

You can control the display of parcel area labels separately for each viewport. A parcel can support more than one area label in addition to the embedded area selection label. With multiple area labels per parcel, you can use layer control to display a different label for each viewport with the correct orientation. You can also manage multiple parcel labels with the Edit Parcel Properties Dialog Box (page 2008).

You can add segment labels automatically when you create a parcel. See Default Settings for New Parcels (page 813).

**To add parcel area labels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels .

2   In the Add Labels dialog box, for Label Type, select Area.

3   The current site is displayed and the following prompt is displayed:

   `Select parcel area selection label or [STart point/Polylines/All/Site]:`

4   Do one of the following:

   ■   Select a start and end point to draw a path through the desired parcel.

- Select **Polylines**, to select a polyline that crosses through the desired parcels. The polyline prompt will repeat until you hit **Enter** to end the command.

- Select **All**, to open a Select Object (Site) dialog box with which to select another available site. After selecting a site, start back at Step 2.

- Select **Site**, to open a Select Object (Site) dialog box with which to select another available site. After selecting a site, start back at Step 2.

5 While you select parcels, the dialog box remains open. You can return to it to change styles or settings for the next sequence of commands. After changing a label style, click Add. If you do not click Add before continuing to click segments, the style change will be ignored.

6 When you finish adding area labels, click Close.


## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels

Menu

Parcels menu ➤ Add Parcel Labels ➤ Add Parcel Labels

Command Line

AddParcelAreaLabel


# Replacing Parcel Area Labels

Use the Replace Area command to easily replace parcel area labels.

**To replace parcel area labels**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels .

2 In the Add Labels dialog box, for Label Type, select Replace Area.

3 You are prompted to select a parcel area label. It will replace that label with a new one using the current Area label style. If the area label style has already been added to the selected parcel a message will be displayed and the label will not be replaced.

While you select parcels, the dialog box remains open. You can return to it to change styles or settings for the next sequence of commands. After changing a label style, click Add. If you do not click Add before continuing to click segments, the style change will be ignored.

4 When you finish replacing area labels, click Close.


## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Parcel ➤ Add Parcel Labels

Menu

Parcels menu ➤ Add Parcel Labels ➤ Add Parcel Labels

Command Line

AddLabels

## Deleting Parcel Segment Labels

You can select and delete parcel segment labels.

**To delete a parcel segment label**

1   Click to select a parcel segment label.

   If you have selected the label correctly, the label text and its anchor point are highlighted.

2   Hit delete.

### Quick Reference

Object Selection

Click label

# Hiding or Showing Parcel Labels

Hide or show labels of any type by setting the Visibility property of their label styles.

For information about setting the Visibility property of label styles, see Editing Parcel Label Styles (page 834). For information about applying label styles, see Applying Parcel Label Styles (page 837).

You can set the Visibility property globally, and in that way hide or show many parcel labels at the same time. See Parcel Label Settings (page 832).

If you do not want site parcels to have area labels, assign an area label style of <none>.

## Hiding or Showing Site Parcel Area Labels

In site parcels, hide or show area labels in two ways.

Hide these labels by choosing a site area label style of <none>. As with other label types, you can also set the Visibility property of their label style to either hide or show them.

There are two types of area labels. The area selection label that is automatically created upon parcel creation cannot be deleted. You must use Ctrl-click to select it. Additional area labels that are added after the parcel is created are independent objects that can be deleted.

**To hide or show a site parcel area label**

1   In Toolspace, on the Prospector tab, expand the Sites collection.

2   Right-click a Parcels collection and click Properties.

3   In the Site Parcel Properties dialog box, click Composition. Select one of the following settings for Site Area Label Style:

**4** ■ To hide labels, set the style to *<none>*, or to a style with Visibility set to False.

■ To show labels, set the style to one with Visibility set to True.

### Quick Reference

Toolspace Shortcut Menu

Prospector: Right-click Site ➤ Parcels ➤ Properties

Dialog Box

Site Parcel Properties (page 2017)

# Parcel Table Styles

Manage parcel table styles in the Table Styles collection in Toolspace, on the Settings tab.

Parcel table styles govern how parcel tables are displayed in a drawing. You can create, edit, or delete parcel table styles.

Create an entirely new table style, or create a table style based on an existing style. When creating parcel tables, you can select general line/curve label types and/or parcel segment labels. You can label parcel segments that use line/curve label styles. Line/Curve label styles can be applied to lines, curves, feature lines, and polyline segments.

## Creating Parcel Table Styles

Create parcel table styles in the Table Styles collection in Toolspace, on the Settings tab.

**To create a parcel table style**

**1** In Toolspace, on the Settings tab, expand Parcel ➤ Table Styles.

**2** Right-click a table type: Area, Line, Curve, or Segment and click New.

**3** In the Table Style dialog box, specify a name for the new style and define the style settings.

### Quick Reference

Toolspace Shortcut Menu

Settings: Table Styles ➤ right-click Area, Line, Curve, or Segment ➤ New

Dialog Box

Table Style (page 2437)

## Editing Parcel Table Styles

Edit parcel table styles in the Table Styles collection in Toolspace, on the Settings tab.

**To edit a parcel table style**

**1** In Toolspace, on the Settings tab, expand Parcel ➤ Table Styles.

**2** Expand a table type: Area, Line, Curve, or Segment.

**3** Right-click the table style that you want to edit and click Edit.

**4** In the Table Style dialog box, modify style settings.

### Quick Reference

Ribbon

Select the table. Click Table tab ➤ Modify panel ➤ Table Properties drop-down ➤ Edit Table Style ⊞

Toolspace Shortcut Menu

Settings: Right-click Parcel table style ➤ New

Dialog Box

Table Style (page 2437)

## Creating Parcel Table Styles Based on Other Styles

Create a parcel table style based on another style in the Table Styles collection in Toolspace, on the Settings tab.

**To create a parcel table style based on another style**

**1** In Toolspace, on the Settings tab, expand Parcel ➤ Table Styles.

**2** Expand a table type: Area, Line, Curve, or Segment.

**3** Right-click the table style that the new style will be based on and click Copy.

**4** In the Table Style dialog box, specify a name for the new style.

**5** Modify style settings.

### Quick Reference

Toolspace Shortcut Menu

Settings: Right-click Parcel table style ➤ Copy

Dialog Box

Table Style (page 2437)

## Deleting Parcel Table Styles

Delete parcel table styles in the Table Styles collection in Toolspace, on the Settings tab.

**To delete a parcel table style**

**1** In Toolspace, on the Settings tab, expand Parcel ➤ Table Styles.

**2** Expand a table type: Area, Line, Curve or Segment.

**3** Right-click the style that you want to delete and click Delete.

### Quick Reference

Toolspace Shortcut Menu

Settings: Right-click Parcel table style ➤ Delete

## Applying Parcel Table Styles

A style is applied to each parcel table in the creation process. You can change this style at any time.

For information about applying table styles in the creation process, see Adding Parcel Tables (page 845).

**To replace a parcel table style**

1    Click a parcel table in the drawing.

2    Right-click and click Table Properties.

3    In the Table Properties dialog box, replace the table style.

### Quick Reference

Ribbon

Select the parcel table. Click Table tab ➤ Modify panel ➤ Table Properties drop-down ➤ Table Properties

Object Shortcut Menu

Parcel table object ➤ Table Properties

Dialog Box

Table Properties (page 2443)

# Adding Parcel Tables

Use parcel tables to consolidate information about the parcels and parcel segments in your drawing.

Tables provide an alternative to labels for information management. Labels can be hard to read if your parcels are small and the drawing is crowded with other data. Tables keep the information organized and separate from the drawing, but cross-referenced by small labels known as tags.

When parcel or segment information is displayed in table rows, table tags uniquely identify the parcels or segments in the drawing and in the initial cells of table rows. For example, if a line segment is labeled L234, its corresponding table row is also labeled L234.

In AutoCAD Civil, parcel tables support general line and curve label types. When creating parcel tables, you can select general line and curve label types and/or parcel segment labels.

For more information about labels, tables, and table tags, see Parcel Labels and Tables (page 831) and Parcel Label Styles (page 833).

Line/curve segments must be labeled prior to adding a table to the drawing. When you add the table, the labels are converted to tags.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. The procedures included in this topic explain how to work with parcel tables. For general information about modifying tables, see Modifying Tables (page 1567).

## Adding Tables of Parcels

A table of parcels, also known as an area table, displays the data for each parcel in a separate row.

Parcel area tables can include data such as parcel perimeter, segment bearings, and street address, depending on the table style.

**To add a table of parcels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Area  .

2   In the Table Creation dialog box, specify settings as needed, except for the Selection settings.

3   To select parcels for the table, for Selection, do one of the following:
    ■   In the Label Style Name list, select one or more area label styles. When you select an area label style, you automatically select all parcels using that style.

    ■   Click the parcel selector button. Click the text of one parcel label after another. When you finish, press Enter.

4   Click OK.
    The upper-left corner of the new table is attached to your pointer.

5   Click in the drawing to set the location for the new table.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Area 

Menu

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Area

Dialog Box

Table Creation (page 2441)

## Adding Tables of Parcel Lines

A table of parcel lines displays the data for each line in a separate row.

For each line that you want to include in the line table, the Table Tag element of its label style must have the Used In property set to Tag Mode. Also, the Display Mode property of the label style must be set to Tag. See Editing Parcel Label Styles (page 834).

**To add a table of parcel lines**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Line  .

2   In the Table Creation dialog box, specify settings as needed, except for the Selection settings.

3   To select segments for the table, for Selection, do one of the following:
    ■   In the Label Style Name list, select one or more segment label styles or general line/curve label styles. When you select a segment label style, you automatically select all segments using that style.

    ■   Click the segment selector button. Click one segment after another. When you finish, press Enter.

**4** Click OK.

The upper-left corner of the new table is attached to your pointer.

**5** Click in the drawing to set the location for the new table.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Line ⊞

Menu

Parcels ➤ Tables ➤ Add Line

# Adding Tables of Parcel Curves

A table of parcel curves displays the data for each curve in a separate row.

For each line that you want to include in the line table, the Table Tag element of its label style must have the Used In property set to Tag Mode. Also, the Display Mode property of the label style must be set to Tag. See Editing Parcel Label Styles (page 834).

**To add a table of parcel curves**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Curve ⊞ .

**2** In the Table Creation dialog box, specify settings as needed, except for the Selection settings.

**3** To select segments for the table, for Selection, do one of the following:

- In the Label Style Name list, select one or more segment label styles or general line/curve label styles. When you select a segment label style, you automatically select all segments using that style.

- Click the segment selector button. Click one segment after another. When you finish, press Enter.

**4** Click OK.

The upper-left corner of the new table is attached to your pointer.

**5** Click in the drawing to set the location for the new table.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Curve ⊞

Menu

Parcels ➤ Add Tables ➤ Add Curve
Parcels menu ➤ Add Tables

# Adding Tables of Parcel Segments

A table of parcel segments displays the data for each segment in a separate row.

For each segment that you want to include in the segment table, the Table Tag element of its label style must have the Used In property set to Tag Mode. Also, the Display Mode property of the label style must be set to Tag. See Editing Parcel Label Styles (page 834).

**To add a table of parcel segments**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Segment ▦.

2   In the Table Creation dialog box, specify settings as needed, except for the Selection settings.

3   To select segments for the table, for Selection, do one of the following:
    ■   In the Label Style Name list, select one or more segment label styles. When you select a segment label style, you automatically select all segments using that style.

    ■   Click the segment selector button. Click one segment after another. When you finish, press Enter.

4   Click OK.
    The upper-left corner of the new table is attached to your pointer.

5   Click in the drawing to set the location for the new table.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Parcel ➤ Add Segment ▦

Menu

Parcels ➤ Add Tables ➤ Add Segments

## Deleting Parcel Tables

Delete parcel tables by clicking in the drawing.

**To delete a parcel table**

1   Click a table title or a table column title in the drawing. The entire table is selected.

2   Right-click in the drawing and click Erase.

## Quick Reference

Object Shortcut Menu

Parcel table object ➤ Erase

## Deleting Parcel Table Rows

Delete parcel table rows by clicking in the drawing.

**To delete a parcel table row**

1   Click the table row you want to delete. The row is selected.

2   Right-click in the drawing and click Erase.

**Quick Reference**

Object Shortcut Menu

Parcel table row object  ➤  Erase

# Parcel Numbering and Naming

Parcels are numbered automatically when you create them. By default, parcel names are based on the parcel style name template (page 1806). You can manually renumber and/or rename them later.

Area parcels are numbered automatically in creation order, beginning with the Parcel: Next Automatic Area Counter in the Site Properties dialog box. When you renumber parcels, numbering begins with the Parcel: Next Manual Area Counter. The same concept applies to segment tags.

Tags are the label elements used as key values in parcel tables.

When numbering automatically, the increment is always 1. When numbering manually (renumbering), you can select the increment when beginning the renumbering process.

# Defining Parcel Numbering Rules

Define numbering rules for parcels, line segments, and curve segments in the Site collection in Toolspace, on the Prospector tab.

For each of these elements, you can specify the next automatic counter for numbering automatically, and the next manual counter for numbering manually (renumbering).

As numbering proceeds, the next-counter values update automatically. These values are properties of the site, and are retained from one session to the next.

**To define parcel numbering rules**

1  In Toolspace, on the Prospector tab, expand Sites.

2  Right-click a site and click Properties.

3  In the Site Properties dialog box, click the Numbering tab.

4  On the numbering tab, specify numbering rules.

**Quick Reference**

Toolspace Shortcut Menu

Prospector: Right-click a site  ➤  Properties

Dialog Box

Site Properties (page 2325)

# Numbering Area Parcels Automatically

Parcels are numbered automatically when you create them.

To display the numbered tags, you must set the Used In property of the Table Tag of the label style to Tag Mode. Also, the Display Mode property of the label style must be set to Tag. See Editing Parcel Label Styles (page 834).

Although this is not the only combination of label style properties that displays the numbered tags, it is the required combination for creating a segment table. A segment table is required for renumbering segments. See Numbering Parcel Segments (page 852).

**To number area parcels automatically**

➤ To number parcels automatically, create the parcels. They will be numbered automatically in creation order. For more information, see Creating Parcels (page 812).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Parcel Creation Tools

Menu

Parcels menu ➤ Create Parcel By Layout

Command Line

CreateParcelByLayout

Dialog Box

Create Parcels by Layout (page 2005)

Dialog Box

Parcel Layout Tools (page 2009)

Ribbon

Home tab ➤ Create Design panel ➤ Parcel drop-down ➤ Create Parcel From Objects

Menu

Parcels menu ➤ Create Parcel From Objects

Command Line

CreateParcelFromObjects

Dialog Box

Create Parcels from Objects (page 2005)

# Renumbering Parcels

Renumber area parcels after they have been created.

Parcels can be renumbered based on a starting number and increment value; they can be renamed based on a name template.

**To renumber parcels**

1 Select the parcel. Click Parcel tab ➤ Modify panel ➤ Renumber/Rename.

2 In the Renumber/Rename Parcels dialog box, specify the site you want to work with and click Renumber.

3 Specify a starting number and an increment value.

**4** Optionally, click Use Name Template In Parcel Style to ensure that the name template is used to resolve the parcel name when renumbering.

**5** Click OK.

**6** Click in the drawing to select parcels using the selection jig (similar to the AutoCAD FENCE command, which selects all objects that cross the selection fence). As each parcel is selected, a temporary line is drawn and parcels faces are highlighted. When you finish picking parcels, the line disappears, enabling you to repeat the selection process for another set of parcels.

The parcel numbers, which are displayed in the parcel area labels, are updated after you exit the command.

**7** When you finish renumbering, press Enter.

### Quick Reference

Ribbon

Select the parcel. Click Parcel tab ➤ Modify panel ➤ Renumber/Rename 

Menu

Parcels menu ➤ Edit Parcel ➤ Renumber/Rename Parcels

Command Line

EditParcelNumbers

Dialog Box

Renumber/Rename (page 2019)

# Renaming Parcels

Rename area parcels after they have been created. The parcels can be renamed automatically based on the name template.

**To rename parcels**

**1** Select the parcel. Click Parcel tab ➤ Modify panel ➤ Renumber/Rename .

**2** In the Renumber/Rename Parcels dialog box, specify the site you want to work with and click Rename.

**3** Specify a name by manually entering a string or click  to open the Name Template dialog box (page 1806) where you can update the default parcel name, using the name template.

**4** Optionally, click Use Name Template In Parcel Style to ensure that the name template is used to resolve the name when renaming.

**5** Click OK.

**6** Click in the drawing to select parcels using the selection jig (similar to the AutoCAD FENCE command, which selects all objects that cross the selection fence). As each parcel is selected, a temporary line is drawn and parcels faces are highlighted. When you finish picking parcels, the line disappears, enabling you to repeat the selection process for another set of parcels.

The parcel names, which are displayed in the parcel area labels, are updated after you exit the command.

**7** When you finish renaming, press Enter.

### Quick Reference

Ribbon

Select a parcel. Parcel tab ➤ Modify panel ➤ Renumber/Rename

Menu

Parcels menu ➤ Edit Parcel ➤ Renumber/Rename Parcels

Command Line

EditParcelNumbers

Dialog Box

Renumber/Rename (page 2019)

# Numbering Parcel Segments

Number parcel segments either before or after they have been created.

The parcel segments can be numbered automatically with the Table Tag Numbering dialog box based on a starting number and increment value.

Line/curve segments can be renumbered only if they are already labeled with a tag label.

**To number parcel segments**

1   In the Table Tag Numbering Dialog Box (page 1971), specify the starting number and increment values.

2   Create a parcel segment table of the segments that you want to number/renumber if they are not in a table already.

See Adding Tables of Parcel Segments (page 847).

3   Click Parcel tab ➤ Labels & Tables panel Renumber Tags ➤ .

4   Click one segment label after another.

Segment numbers displayed in the labels are updated as you click them. Also, the numbers displayed in the first cells of each row of the segment table are updated.

5   When you finish renumbering segments, click OK. At the command prompt, enter **E** for End.

### Quick Reference

Ribbon

Parcel tab ➤ Labels & Tables panel ➤ Renumber Tags

Menu

Parcel menu ➤ Add Tables ➤ Renumber Tags

Command Line

EditParcelTagNumbers

Dialog Box

Renumbering (page 2019)

# Using User-Defined Property Classifications with Parcels

You can use user-defined Property Classifications to organize and group specific types of user-defined properties.

For more information about User-defined Property Classifications, see User-Defined Property Classifications (page 416).

## Assigning User-Defined Properties to Parcels

Assign a User-Defined Classification to a parcel.

After you create a User-Defined Classification, you can assign it to the Parcels collection using the Site Parcels Properties dialog box (page 2017). The properties within the classification are displayed as columns in the Parcel item list view.

**To assign user-defined properties to a parcel**

1   In the drawing, define the properties within the User-Defined Classification. For more information, see Creating a User-Defined Property Classification (page 417).

2   In Toolspace, on the Prospector tab, click the Parcels collection.

3   Right-click and click Parcel Properties.

4   In the Site Parcel Properties dialog box, click the Composition tab.

5   On the Composition tab, under User-Defined Property Classification, select the User-Defined Classification you want to assign to the Site Parcel.

6   In Toolspace, on the Prospector tab, select the parcel to which you assigned the User-Defined Classification.

7   In the parcel list view window, right-click a column heading. Select the user-defined properties you want to display in the list view window.

> **NOTE** You can arrange the display of columns in the list view by dragging column headings. For more information, see Customizing a List View (page 84).

## Parcels Command Reference

The following table lists parcel commands and briefly describes their functionality. For more information about a command, follow the link in the Description column.

| Command | Description |
| --- | --- |
| AddParcelCurveTable | Adds a parcel curve table (page 847) |
| AddParcelLabels | Adds parcel labels (page 838) |
| AddParcelLineLabel | Adds a parcel segment label, either line or curve (page 839) |
| AddParcelLineTable | Adds a parcel line table (page 847) |
| AddParcelSegmentLabels | Adds parcel segment labels (page 839) |
| AddParcelSegmentTable | Adds a parcel segment table (page 847) |

| Command | Description |
| --- | --- |
| AddParcelTable | Adds a parcel table (page 846) |
| CreateParcelByLayout | Creates a parcel by layout (page 815) |
| EditParcel | Edits parcels by layout (page 820) |
| EditParcelNumbers | Renames or renumbers parcels (page 850) |
| EditParcelTagNumbers | Edits parcel tag numbers (page 852) |
| ExportParcelAnalysis | Exports a parcel inverse or mapcheck report (page 830) |
| CreateParcelFromObjects | Creates a parcel from AutoCAD objects (page 814) |
| CreateParcelROW | Creates an automatic right-of-way (page 822) |
| ParcelPropertiesEdit | Edits parcel properties (page 825) |

# Alignments

# 23

Alignment objects can represent road centerlines, pipe networks, and other construction baselines. Creating and defining a horizontal alignment is one of the first steps in roadway, railroad, or site design. You can draw the alignment geometry as a polyline, and then create the named alignment from that geometry. For greater control, you can create an alignment object using the Alignment Layout Tools. You can also make edits to alignments using grips, or the commands on the Alignment Layout Tools toolbar, while automatically maintaining tangency between the alignment components.

For information on recommended workflows when working with alignments, see the *AutoCAD Civil Workflow Guide*.

## Understanding Alignment Objects

You create alignments as a combination of lines, curves, and spirals that are viewed as one object.

Alignments can be stand-alone objects or the parent object of profiles, sections, and corridors. If you edit an alignment, the changes are automatically reflected in any related objects.

## Overview of Alignments

In AutoCAD Civil, you create alignments as objects.

Use the Prospector and Settings tabs to manage alignments in Toolspace.

Alignments are listed in either the Alignments collection or a Site collection in the Prospector tab. From these collections, you can change alignment properties, generate reports, and export LandXML. See Alignment Collection (Prospector Tab) (page 856) for more information.

Use the Settings tab in Toolspace to manage the details of alignment object styles, label and table styles, and command settings.

Control the visual appearance of alignment components and labels through styles. You can create specific styles to use in the various phases of a project. For more information, see Alignment Styles (page 885).

When you create an alignment, you specify a style for the alignment object and styles for the alignment labels. In the Alignment Labels Dialog Box (page 1739), you can create groups of styles for alignment labels, and then save those labels as a label set to use with other alignments.

## Alignment Types

When creating an alignment, assign it a type of centerline, offset, miscellaneous, or curb return.

You can use these alignment types to categorize your data based on alignment function. Select the centerline type for a road centerline, or the miscellaneous type for other uses, such as a utility conduit. These two types are independent objects.

Offset alignments (page 2479) and curb return alignments (page 2471) can also be independent, but are often dynamically linked to other alignments. Both offset alignments and curb returns can be created automatically as components of an intersection. Offsets can also be used in widenings.

**See also:**

■ Creating Alignments (page 887)

■ Widenings (page 896)

# Alignment Object Relationships

Alignment objects can establish spatial parameters for other objects, including profiles, sections, corridors, intersections, and pipe networks.

If an alignment exists in a site, its geometry interacts with that of other objects in the site, such as parcels and feature lines. For example, an alignment subdivides any parcels it intersects within the site. Any lot lines or feature lines that cross an alignment assume the same elevation at the intersection point. The elevation is set by whichever object is last edited. To prevent an alignment from interacting with other objects in a site, set the site property to <None> or move the alignment to a separate site. For more information, see Alignment and Site Interaction (page 721).

# Alignment Collection (Prospector Tab)

Use the Alignment collection in the Prospector tree to access the alignments in a drawing.

As alignment objects are created, they are displayed as named alignments, organized by type, in one of two collections:

■ The top-level Alignments collection. When alignments are in the top-level Alignments collection, they do not interact with parcels contained in Site collections. For more information, see Alignment and Site Interaction (page 721).

■ A project Site collection. An alignment object can exist in only one Site. When an alignment is in a Site collection, its geometry interacts with that of other objects in the site, such as parcels and feature lines.
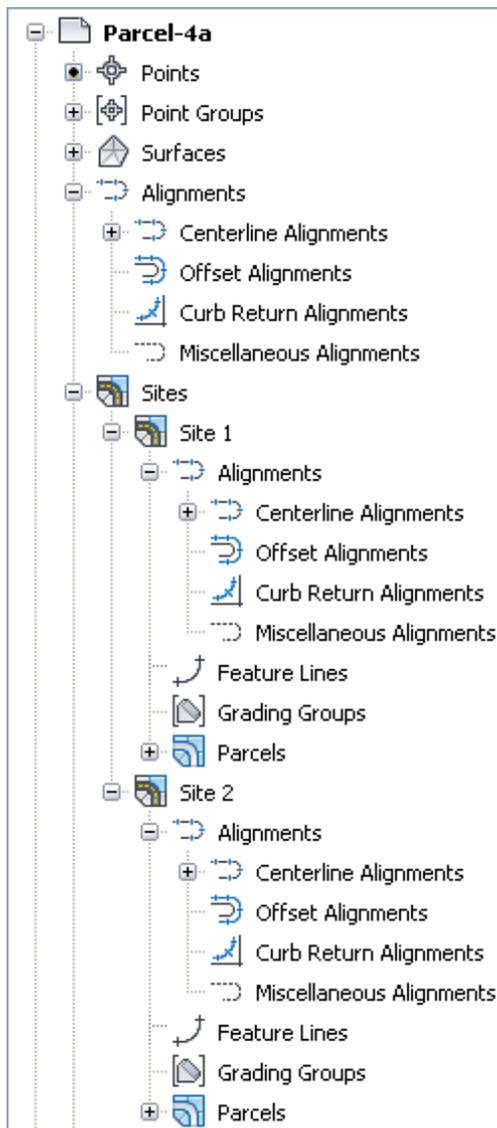
**Sample alignments collections on the Toolspace Prospector tab**

Expand an Alignment collection to view the names of the alignments. Select an Alignment collection to display a detailed tabular list of Alignment information at the bottom of the Prospector tab. For more information, see The Toolspace Item View (page 82).

Right-click an individual alignment in the Alignment collection to:

■ View and edit the alignment properties

■ Move or copy an alignment to either another Site or the top-level Alignments collection

■ Zoom or pan to the alignment

■ Export the alignment to LandXML

Expand an individual alignment name to display profiles, profile views, and sample line groups that are associated with the alignment. These items are also displayed in the tabular list view at the bottom of the Prospector tab.

| For more information about... | See... |
| --- | --- |
| Profiles | Profiles Collection (Prospector Tab) (page 1022) |
| Profile Views | Profile Views Collection (Prospector Tab) (page 1022) |

# Alignment Collection (Settings Tab)

Manage alignment settings, alignment styles, alignment label styles, alignment table styles, and alignment command settings in Toolspace, on the Settings tab.

Right-click the Alignment collection to:

- Edit the alignment feature settings.

- Edit the default alignment label styles.

- Refresh the display of the settings tree.

Expand the Alignment collection to display and edit the alignment styles, label and table styles, and command settings.

**See also:**

- Alignment Settings (page 860)

- Alignment Labels and Tables (page 971)

## The Alignment Styles Collection

Use the Alignment Styles collection in the Settings tree, which lists alignment styles that are available in a drawing, to create an alignment style.

Expand the Alignment Styles collection to display a list of available alignment styles.

Right-click Alignment Style ➤ New to create an alignment style.

---

**NOTE** When you click New in a collection, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a style from an existing style, right-click the style. Click Copy, and save the style with a new name.

---

Right-click an existing alignment style to display the shortcut menu commands:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Change the selected style. |
| Copy | Make a copy of the selected style. |
| Delete | Delete the selected style. |
| Refresh | Update the current view. |

**See also:**

■ Alignment Styles (page 885)

## The Alignment Design Checks Collection

Use the Alignment Design Checks collection to manage the user-defined checks that verify alignment design parameters.

Alignment design checks (page 2472) can be created for Line, Curve, Spiral, or Tangent Intersection entity types. To apply a design check to an alignment, you must add it to a design check set (page 2472).

To edit, create, or copy a design check, right-click an existing design check. Click a command on the shortcut menu:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Edit the design check. |
| Copy | Copy the design check. |
| Delete | Delete the selected design check. You can only delete design checks that are not referenced by a design check sets. |
| Refresh | Update the current view. |

**See also:**

■ Alignment Design Checks (page 871)

## The Alignment Label Styles Collection

Use the Alignment Label Styles collection to override default alignment label settings on the drawing level and create new styles.

Alignment labels are independent objects that are managed with the AutoCAD Properties palette. At the feature level, you can override the default label settings you established at the drawing level. You can edit or create a label style for a specific label type.

**NOTE** When you click New in a collection, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a style from an existing style, right-click the style. Click Copy, and save the style with a new name.

To edit, create, or copy a label style, right-click an existing alignment label style. Click a command on the shortcut menu:

| Select this command... | If you want to... |
| --- | --- |
| Edit | Edit the current label style. |
| Copy | Copy the style. |
| Delete | Delete the selected style. You can only delete styles that are not referenced by other objects or label sets. |
| Refresh | Update the current view. |

**See also:**

- Alignment Label Styles (page 971)

## The Alignment Table Style Collection

Use the Alignment Table Style collection to create a table style or to modify an existing table style.

To edit, copy, or delete an alignment table style, right-click an alignment table style. Click a command on the shortcut menu.

---

**NOTE** When you click New in a collection, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a style from an existing style, right-click the style. Click Copy, and save the style with a new name.

---

To edit, create, or copy a table style, right-click an existing alignment label style. Click a command on the shortcut menu:

| Select this command... | If you want to... |
|---|---|
| Edit | Edit the current table style. |
| Copy | Copy the style. |
| Delete | Delete the selected style. You can only delete styles that are not referenced by other objects or label sets. |
| Refresh | Update the current view. |

**See also:**

- Alignment Table Styles (page 986)

## The Alignment Command Collection

Control the settings for a specific alignment command using the Alignment Commands collection.

When you change settings for a command in the collection, the changes affect that command only.

Expand the Alignment  Commands collection, on the Settings tab, to display a list of commands for an alignment.

Right-click a command name and click Edit Command Settings to open the Edit Command Settings dialog box for the command. Change the settings as needed.

**See also:**

- Alignment Command Reference (page 1016)

# Alignment Settings

You can use alignment settings to specify the default behavior for alignment commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Settings tree. Control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

Use the Settings tree Alignment collection shortcut menu to establish defaults for all alignment commands. You can change alignment-specific settings at this level, such as Station Indexing and also override the drawing ambient settings.

Use the Commands collection under the Alignment collection to change alignment settings for a specific command. You can change alignment-specific settings at this level, or override the drawing ambient settings.

The topics in this section describe only those settings that affect alignment-related commands. The topics in this section do not cover the drawing ambient settings, which are also displayed in the Alignment Settings dialog box.

**See also:**

■ Specifying Drawing Settings (page 63)

**To edit the alignment feature settings**

1  In Toolspace, on the Settings tab, right-click the Alignment collection ➤ Edit Feature Settings.

2  In the Alignment Settings dialog box, expand the display for each setting. Click a cell in the Value column and enter a new value or specify a different style.

3  Click Apply to accept the changes and continue working in the dialog box or click OK to accept the changes and close the dialog box.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Alignment ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings - Alignment Dialog Box (page 1767)

# Curve and Spiral Settings

Enter the curve settings you want to use when creating alignments using the Tangent-Tangent (With Curves) command.

You can specify default settings for curves and spirals by editing the settings in the CreateAlignmentLayout command.

While you can select various combinations of Spiral In, Curve, and Spiral Out, the A value of the spirals is controlled by the radius of the curve. The length of the spirals is controlled by changing the A Value.

## Specifying Curve and Spiral Settings

Specify default spiral type and values in the Curve and Spiral Settings dialog box or by editing the Curve and Spiral Settings for the CreateAlignmentLayout command.

The parameter values in the Curve and Spiral Settings dialog box apply only to the curves and spirals created using the Tangent-Tangent with Curves command in the current alignment. You can specify default settings for all curves and spirals by editing the settings in the CreateAlignmentLayout command.

**To change the default Curve and Spiral Settings for all instances**

1  On the Settings tab in Toolspace, expand the Alignments collection and then expand the Commands collection.

**2** Right-click CreateAlignmentLayout ➤ Edit Command Settings.

**3** In the Edit Command Settings-CreateAlignmentLayout dialog box, expand the Curve and Spiral Settings property.

**4** Specify the settings, click Apply and click OK.

> **NOTE** To save these command settings at the drawing level, on the Ambient Setting tab (page 68) select Yes in the value column for Save Command Changes To Settings.

**To change the Curve and Spiral Settings used by the tangent-tangent with curves method**

**1** Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools .

**2** In the Create Alignment-Layout (page 1763) dialog box, enter creation information.

**3** On the Alignment Layout Tools toolbar, click  Curve And Spiral Settings.

**4** In the Curve and Spiral Settings (page 861) dialog box, enter curves values.

> **NOTE** The spiral type you specify in the Curve and Spiral Settings dialog box is used in all spiral commands on the Alignment Layout Tools toolbar.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools 

Menu

Alignments menu ➤ Create Alignment By Layout

Alignment Layout Tools Toolbar Icon

 Curve And Spiral Settings

Command Line

EditAlignment

Dialog Box

Alignment Layout Tools (page 1747)

## Specifying a Curve Group Index

Use each index to identify the entities that form part of a complex alignment.

The Curve Group Index identifies a curve group, such as a spiral-curve-spiral group, within an alignment.

The Curve Group Sub-Entity Index identifies each entity within a curve group.

When you label an alignment, you can use these values for alignment identification.

**To specify a curve group index**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

**2** On the Alignment Layout Tools toolbar, click  Alignment Grid View.

**3** In the Alignment Entities Vista (page 1735), enter values for the Curve Group Index and the Curve Group Sub-Entity Index.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 

Menu

Alignments menu ➤ Edit Alignment Geometry

Alignment Layout Tools Toolbar icons

 Alignment Grid View

Dialog Box

Alignment Entities Vista (page 1735)

# Alignment Properties

You can use the Alignment Properties dialog box to assign a name, style, and other properties to the alignment.

You can adjust the style, stationing, design criteria, and superelevation parameters of a selected alignment. You can also view basic information about profiles and profile views that have been created for the current alignment.

Use the Superelevation tab to define the specifications for Superelevation Regions that are associated with curve groups in the alignment. Superelevation is used with the corridor model to determine the slopes of travel lanes and shoulders at each station in the model.

The Alignment Properties dialog box has the following tabs:

■ **Information**. Specifies name, description, and object style.

■ **Station Control**. Displays alignment length and raw stationing as well as options to change the station reference point, and apply station equations.

■ **Masking**. Specifies one or more regions of the alignment that can be suppressed from display.

■ **Design Criteria**. Specifies design speed options to select points in the drawing where you want to assign travel speeds to aid in design and labeling. When you assign a design speed to an alignment station, that speed is applied to all subsequent points on the alignment, unless another speed is applied at some point. This value also displays in the Superelevation Specification dialog box.
This tab also contains the options related to criteria-based design (page 2471). This optional feature allows you to specify the design criteria file, default criteria, and design check set for the alignment.

■ **Profiles**. Displays information for profiles associated with the alignment.

■ **Profile Views**. Displays information for profile views associated with the alignment.

■ **Superelevation**. Displays the data for each superelevation region after calculation.

■ **Offset Parameters**. If the alignment is a dependent offset, this tab displays properties of the offset.

■ **Curb Return Parameters**. If the alignment is a curb return, this tab displays data about the two parent alignments, offset values, and update mode.

**NOTE** Alignment label properties are controlled using the Alignment Labels Dialog Box (page 1739).

**See also:**

■ Alignment Properties Dialog Box (page 1752)

**To change alignment name, description, or style**

1 Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties  .

> **NOTE** The tab most recently opened in the Alignment Properties dialog box is displayed by default.

2 In the Alignment Properties dialog box, click the Information tab (page 1752).

3 Under Name, enter a name for the alignment.

4 Under Description, enter an optional description for the alignment.

5 In the Object Style list, select an existing alignment style.

6 Click the Style Detail button to open the Style Detail dialog box. Preview the alignment style.

**To change alignment Station Control**

1 Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties  ..

> **NOTE** The tab most recently opened in the Alignment Properties dialog box is displayed by default.

2 In the Alignment Properties dialog box, click the Station Control (page 1752) tab.

3 Under Reference Point, click  . The Alignment Properties dialog box closes.

> **WARNING** Changing the reference point location or the reference station value removes all station equations and design speeds and can adversely affect objects and data already created from the alignment. Click OK to continue or Cancel to abort this operation. Use Station Equations to change stations after you create Profiles and Cross Sections.

4 In the drawing, pick a point where you want to establish the XY coordinates for the Reference Point. The Alignment Properties dialog box is displayed again.

5 In the Reference Point box, under Station, enter a starting station value.

6 In the Station Equations box, click  . The Alignment Properties dialog box closes.

7 In the drawing, pick the point for the first station equation. The Alignment Properties dialog box is displayed again.

8 Repeat Steps 6 and 7 to add more station equations.

9 By default Station Ahead and Raw Station Back have the same value. Click the Station Ahead cell and enter a new value.

10 Click the Increase/Decrease cell to determine whether the station values from the Station Equation increases or decreases from the Station Ahead value.

**To mask an alignment**

1  Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties 🔲 .

2  In the Alignment Properties dialog box, click the Masking (page 1753) tab.

3  Click ➤ Add Masking Region.

4  Specify the start station for the mask by clicking along the alignment, entering a numeric station value, or clicking Lock to Start in the masking table.

5  Specify the end station for the mask by clicking along the alignment, entering a numeric station value, or clicking Lock to End in the masking table.

6  Optionally, enter notes about the mask in the Comment column.

**TIP** Use the Mask check box to temporarily enable or disable a mask without removing it from the list of masks.

**To change alignment design speeds or criteria**

1  Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties 🔲 .

**NOTE** The tab most recently opened in the Alignment Properties dialog box is displayed by default.

2  In the Alignment Properties dialog box, click the Design Criteria (page 1754) tab.

3  Click 🔲 .

A new row is displayed in the table.

4  In the Start Station column, enter the station at which the design speed begins.

5  For Design Speed, enter the design speed.

The default Design Speed value is specified in the alignment feature settings.

6  Repeat Steps 3 through 5 to add more design speeds.

An alignment can have unlimited numbers of design speeds, but only one speed at any given station. When a station has a design speed, that speed applies to subsequent stations until either the next station at which a design speed is set, or the alignment end station. If a single entity has multiple speeds assigned to it, the highest speed is used to validate that entity against the design criteria. The lower speed values are ignored.

**NOTE** To calculate superelevations on the Superelevation tab, you must specify design speeds.

7  On the right side of the dialog box, modify the design criteria as desired.

8  Click OK.

**NOTE** If design criteria has been associated with the alignment, the entire alignment design is validated when you click either Apply or OK.

**To view Profiles associated with the alignment**

**1** Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties .

> **NOTE** The tab most recently opened in the Alignment Properties dialog box is displayed by default.

**2** Click the Profiles tab. For more information, see Profile Data Tab (Profile Properties Dialog Box) (page 2174).

**To view Profile Views associated with the alignment**

**1** Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties .

> **NOTE** The tab most recently opened in the Alignment Properties dialog box is displayed by default

**2** Click the Profile View tab. For more information, see Profiles Tab (Profile View Properties Dialog Box) (page 2179).

### Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties 

Prospector Shortcut Menu

Right-click alignment ➤ Properties

Object Shortcut Menu

Right-click <alignment object> ➤ Alignment Properties

Command Line

EditAlignmentProperties

Dialog Box

Alignment Properties Dialog Box (page 1752).

# Criteria-Based Alignment Design

Use the criteria-based design feature to apply agency-specific standards to an alignment.

The criteria-based design feature provides the ability to verify that your alignment design meets the minimum standards required by your local agency.

When you use the criteria-based design feature, you can select a design criteria file (page 2472), from which you can specify the superelevation attainment method and minimum radius and transition (spiral) length tables. When you lay out the alignment, the appropriate minimum values specified in the design criteria file are displayed on the command line. You can either accept the default minimum value for a given sub-entity, or specify a new value.

Minimum transition length values are automatically generated using the specified curve radius. If there is no curve associated with the spiral, then the minimum value is the smallest acceptable spiral radius.

If the design parameters for a sub-entity violate the minimum values established in the design criteria file, a warning symbol appears both on the sub-entity in the drawing window and next to the violated value in the Alignment Entities Vista (page 1735) and Alignment Layout Parameters Window (page 1741). When you hover the cursor over a warning symbol, a tooltip displays which standard has been violated and how to correct the violation. The display of the warning symbol is controlled by the alignment style.

Some alignment design criteria are not available in table form in the design criteria file. For these criteria, you can define design checks (page 2472) to validate design standards. To apply a design check to an alignment, you must add it to a design check set (page 2472).

**NOTE** Compound spiral length is not validated against the design criteria file. You must use design checks to validate compound spiral length.

You can generate a report that documents validations or violations in the alignment design. The design criteria report identifies whether each sub-entity within a given station range violates or meets the appropriate design criteria and design checks.

# Design Criteria Files

A design criteria file contains the standards tables that specify appropriate minimum design standards for alignments or profiles.

The design criteria file (page 2472) contains the minimum design standards for alignment and profile objects. You can customize the design criteria file to support local design standards for such things as superelevation and minimum K values.

Design criteria files containing A.A.S.H.T.O. (page 2467) standards in both metric and imperial units are included with AutoCAD Civil.

If your local agency standards differ from those in the supplied design criteria files, you can use the Design Criteria Editor Dialog Box (page 1843) to customize the file to support your local standards.

**NOTE** If you use a custom a design criteria file, you must save it to a shared server to which all members of your design team have access. If you send a drawing that uses a custom design criteria file, you must send the design criteria file with the drawing.

**Alignment Design Criteria**

A design criteria file can contain the following alignment design criteria:

■ Minimum radius at a given design speed

■ Superelevation attainment method

■ Superelevation rate at a given radius

■ Minimum transition length at a given radius

**NOTE** Compound spiral length is not validated against the design criteria file. You must use design checks to validate compound spiral length.

**Profile Design Criteria**

A design criteria file can contain standards tables for minimum K values at specified design speeds. Minimum K tables for the following distances are available:

■ Stopping Sight Distance

■ Passing Sight Distance

■ Headlight Sight Distance

**See also:**

■
■
■

## Creating a Design Criteria File

Create a customized design criteria file to reflect your local standards.

If you do not want to use or modify the provided design criteria file, you can create a custom design criteria file in either of the following ways:

■ Copy the provided design criteria file. Rename the copy, and then modify the copy to reflect your design standards.

■ Create an empty design criteria file. Add your local standards using the schema that is defined in the design criteria file.

**To copy the provided design criteria file**

**1** Click an alignment or profile in the drawing

**2** Click Alignment tab ➤ Modify panel ➤ Design Criteria Editor ⬚ .

**3** In the Design Criteria Editor Dialog Box (page 1843), click ⬚ Open.

**4** In the Select A Design Criteria File dialog box, select the design criteria file you want to copy. Click Open.

**5** In the Design Criteria Editor Dialog Box (page 1843), click ⬚ Save As.

**6** In the Enter A File Name To Save dialog box, enter a new name for the design criteria file. Click Save.

> **NOTE** Design criteria files are saved in this directory: *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>*. The criteria-based design feature uses the first file found in the directory. To ensure that your new file is used by default, give it a name that places it before all other files in the directory.

**7** In the Design Criteria Editor dialog box, click ⬚ Expand All to expand all nodes in the tree on the left side of the dialog box.

**8** Select the desired ⬚ table.

The table contents are displayed in the grid view on the right side of the dialog box.

**9** Edit contents of the design criteria file as desired.

■ To create new ⬚ tables and ⬚ folders in the tree view, right-click the desired node and use the commands on the shortcut menu.

■ To edit a value, double-click the value.

- To add a row to a table, click ![add icon] .

- To remove a row from a table, click ![remove icon] .

**10** Click Save And Close to save the new design criteria file and close the Design Criteria Editor dialog box.

**See also:**

- Design Criteria Files (page 867)
- Profile Standards in the Design Criteria File (page 1045)
- Superelevation Specification Variables (page 880)
- Examples of Attainment Methods (page 881)

**To create an empty design criteria file**

**1** Click an alignment or profile in the drawing

**2** Click Alignment tab ➤ Modify panel ➤ Design Criteria Editor ![icon] .

**3** In the Design Criteria Editor Dialog Box (page 1843), click ![icon]

**4** New.

An untitled design criteria file is created, with empty Units, Alignments, and Profiles folders in the tree view.

**5** In the tree view, right-click Units. Click either New Metric or New Imperial.

**6** In the grid view, specify values for the units parameters.

**7** In the tree view, right-click either the Alignments or Profiles folder. Click the desired table category.

**8** Right-click the table category folder you created in Step 5. Click the desired table.

**9** Enter a name for the table you created in Step 6.

**10** Edit contents of the table as desired.
- To edit a value, double-click the value.

- To add a row to a table, click ![add icon] .

- To remove a row from a table, click ![remove icon] .

**11** Repeat Steps 5 through 8 to add more tables to the design criteria file.

**12** In the Design Criteria Editor dialog box, click ![save icon] Save.

**13** In the Enter A File Name To Save dialog box, enter a new name for the design criteria file. Click Save.

---

**NOTE** Design criteria files are saved in this directory: *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>*. The criteria-based design feature uses the first file found in the directory. To ensure that your new file is used by default, give it a name that places it before all other files in the directory.

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**14** Click Save And Close to save the new design criteria file and close the Design Criteria Editor dialog box.

**See also:**

■ Design Criteria Files (page 867)

■ Profile Standards in the Design Criteria File (page 1045)

■ Superelevation Specification Variables (page 880)

■ Examples of Attainment Methods (page 881)

## Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Design Criteria Editor

Menu

Click Alignmentsmenu ➤ Design Criteria Editor.

Dialog Box

Design Criteria Editor Dialog Box (page 1843)

## Editing a Design Criteria File

Modify a design criteria file to reflect your local standards.

You use the Design Criteria Editor Dialog Box (page 1843) to edit the design criteria file.

**To edit a design criteria file**

**1** Click an alignment or profile in the drawing

**2** Click Alignment tab ➤ Modify panel ➤ Design Criteria Editor .

**3** In the Design Criteria Editor Dialog Box (page 1843), expand the desired folder in the tree on the left side of the dialog box.

**4** Select the desired table.
The table contents are displayed in the grid view on the right side of the dialog box.

**5** Edit contents of the design criteria file as desired.

■ To create new tables and folders in the tree view, right-click the desired node and use the commands on the shortcut menu.

■ To edit a value in the grid view, double-click the value.

■ To add a row to a table, click .

■ To remove a row from a table, click .

**6** Click Save And Close to save the design criteria file with your changes and close the Design Criteria Editor dialog box.

**See also:**

- Design Criteria Files (page 867)
- Profile Standards in the Design Criteria File (page 1045)
- Superelevation Specification Variables (page 880)
- Examples of Attainment Methods (page 881)

### Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Design Criteria Editor 

Menu

Click Alignments menu ➤ Design Criteria Editor

Dialog Box

Design Criteria Editor Dialog Box (page 1843)

## Alignment Design Checks

Create design checks to verify the design parameters for criteria that are not included in the design criteria file.

The tables contained in the design criteria file (page 2472) contain the most commonly used criteria for alignment design. Other criteria can be checked by using mathematical expressions known as design checks (page 2472).

Design checks are created and managed in the same manner as label expressions. Use the Design Check Editor Dialog Box (page 1845), which is similar to the Expressions Dialog Box (page 1973), to create and edit design checks. To apply a design check to an alignment, you must add it to a design check set (page 2472), which is similar to an alignment label set.

You can create design checks for lines, curves, spirals, and tangent intersections, which are groups of multiple line, curve, and spiral entities.

Independent entities are checked separately from sub-entities in tangent intersection groups. To apply the same design check to both entiity types, create a separate design check for each type. For example, to enforce a minimum radius for all curves, create a curve design check for use with the independent curves and a tangent intersection design check for use with spiral-curve-spiral groups.

Design checks differ from label expressions in that they always return a true or false value. For example, you can create a design check to issue a notification if a tangent is shorter than a specified minimum tangent length.

**See also:**

- Expressions (page 1547)
- Using Alignment Label Sets (page 973)

### Design Check Examples

Following are some examples of design check syntax.

The following examples demonstrate the type of parameters that can be validated using design checks. Use the examples as a basis to create your own custom design checks.

---

**NOTE** Several design checks are included in the AutoCAD Civil templates in the Toolspace ➤ Settings ➤ Alignment and Profile ➤ Design Checks collections.

---

### Minimum Tangent Length

This design check validates that a tangent is a minimum specified length.

| | |
|---|---|
| Name | Tangent Length >= 100 |
| Type | Line |
| Formula | {Tangent Length}>=100 |
| Formula Description | Tangents must be greater than or equal to 100. |

### Minimum Tangent Length at a Design Speed

This design check validates that a minimum tangent length is used at a given design speed.

| | |
|---|---|
| Name | Tangent Length >= 100 @ 45mph |
| Type | Line |
| Formula | IF({Design Speed}>=45,Length>=100,1) |
| Formula Description | If the design speed is greater than or equal to 45, the tangent length must be greater than or equal to 100. <br><br> **NOTE** In the IF function, the ending 1 indicates that if the entity meets both the design speed and length conditions, a true value is returned. A true value means that the entity meets the design check, and no warning is issued. If the entity does not meet one or both of the conditions of the design check, a warning is issued. |

### Design Speed On Curve Radius

This design check validates that a minimum curve radius is used at a given design speed. This design check is relatively complex, and combines several formulas into a single design check.

In this example, the minimum curve radius at eight separate design speeds is validated. When applied to an alignment, the design check checks the curve design speed value, and then validates that the radius is greater than or equal to the specified value.

| | |
|---|---|
| Name | Design Speeds On Curve Radii (Multiple) |
| Type | Curve |
| Formula | IF({Design Speed}>=120,Radius>=750,<br>IF({Design Speed}>=100,Radius>=500,<br>IF({Design Speed}>=80,Radius>=300,<br>IF({Design Speed}>=70,Radius>=200,<br>IF({Design Speed}>=60,Radius>=125,<br>IF({Design Speed}>=50,Radius>=80, |

IF({Design Speed}>=40,Radius>=50,
IF({Design Speed}>=30,Radius>=30,
Radius>= 0))))))))

| Formula De-scription | If the design speed is greater than or equal to one of the specified speed values, the radius must be greater than or equal to the specified radius value for that speed. |
| --- | --- |

**Design Speed On Tangent Length**

This design check validates that the tangent length is within a specified range at a given design speed. This design check is relatively complex, and combines a several formulas into a single design check.

In this example, tangent length at five separate design speeds is validated. When applied to an alignment, the design check checks the tangent design speed value, and then validates that the tangent length is within the specified range.

| Name | Design Speeds On Tangent Lengths (Multiple) |
| --- | --- |
| Type | Line |
| Formula | IF({Design Speed}>=120,(2000>=Length)*(Length>=500), IF({Design Speed}>=100,(2000>=Length)*(Length>=400), IF({Design Speed}>=80,(1500>=Length)*(Length>=350), IF({Design Speed}>=70,(1200>=Length)*(Length>=300), IF({Design Speed}>=60,(1000>=Length)*(Length>=250),Length>=0))))) |
| Formula De-scription | If the design speed is greater than or equal to one of the specified speed values, the tangent length value must be between the maximum and minimum values specified for that speed. |

**See also:**

- Alignment Design Checks (page 871)
- Profile Design Checks (page 1045)

# Creating a Design Check

To create a design check, set up a mathematical formula that uses existing alignment sub-entity properties.

Use the Toolspace Settings tab to create and manage design checks. You must create separate design checks for lines, curves, spirals, and curve groups (tangent intersections). To apply a design check to an alignment, you must add it to a design check set (page 2472).

Independent entities are checked separately from sub-entities in tangent intersection groups. To apply the same design check to both entiity types, create a separate design check for each type. For example, to enforce a minimum radius for all curves, create a curve design check for use with the independent curves and a tangent intersection design check for use with spiral-curve-spiral groups.

**See also:**

- Alignment Design Checks (page 871)

**To create a design check**

**1**  In Toolspace, on the Settings tab, expand the Alignment ➤ Design Checks collection.

**2**  Right-click the entity type collection for which you want to create a design check. Click New.

Alternatively, you can expand the entity type collection for which you want to create a design check, right-click an existing design check, and click Copy.

**3**  In the Design Check Editor Dialog Box (page 1845), enter a name and description for the design check.

> **BEST PRACTICE** The name of the design check should be similar to the formula you enter in the Design Check field. When a design check is violated, only the design check name is displayed, and not the minimum acceptable value. If the formula is in the design check name, it is easier to correct the violation.

**4**  Click Insert Property. Select a property from the list.

The property appears in the Design Check field.

**5**  Add functions, constants, and logical operators to complete the design check.

Because design checks return either a true or false value, they must contain one of the following logical operators: > (greater than), < (less than), >= (greater than or equals), <= (less than or equals), != (not equals), = (equals).

> **NOTE** Do not insert anything into the { } brackets that surround the property fields.

■  Click Insert Function to insert mathematical functions like TAN.

■  Use the calculator buttons or click in the Design Check field and use your keypad to enter numbers, constants, and mathematical operators like **\*** (multiply) or / (divide).

> **NOTE** For a complete list of operators and functions, see Design Check Editor Dialog Box (page 1845).

**6**  Click OK.

> **NOTE** To apply a design check to an alignment or profile, it must be saved in a design check set. For more information, see Creating a Design Check Set (page 875) or Editing a Design Check Set (page 876).

**See also:**

■  Alignment Design Checks (page 871)

■  Design Check Examples (page 871)

## Quick Reference

Toolspace Shortcut Menu

<drawing name> ➤ Alignment ➤ Design Checks ➤ <design check type collection> ➤ <right-click design check in item view>

Dialog Box

Design Check Editor Dialog Box (page 1845)

## Editing a Design Check

After you create a design check, you can edit it from the Settings tab in Toolspace.

Click the appropriate Design Checks collection to display the list of defined design checks in the Toolspace item view. Right-click a design check to access commands.

**See also:**

■ Alignment Design Checks (page 871)

**To edit a design check**

1 In Toolspace, on the Settings tab, select the Design Checks collection that contains the design check that you want to edit.

2 Ensure the Toolspace item view is visible.

3 Right-click a design check in the item view. Click Edit.

4 Edit the design check using the Design Check Editor Dialog Box (page 1845).

### Quick Reference

Toolspace Settings Tab Shortcut Menu

<drawing name> ➤ Alignment ➤ Design Checks ➤ <design check type collection> ➤ <right-click design check in item view>

Dialog Box

Design Check Editor Dialog Box (page 1845)

## Alignment Design Check Sets

Save commonly used combinations of design checks in a design check set, which is then applied to an alignment.

After you create a design check (page 2472), save it with an appropriate combination of other design checks in a design check set. A design check set can be applied to an alignment either during the alignment creation process, or through the properties of an existing alignment.

A design check set consists of design checks for a combination of design check types. For example, you can create a design check set that consists of separate design checks for minimum and maximum A values for spirals, minimum radii for curves, and minimum spiral transition lengths.

Design check sets are created and managed in the same manner as label sets. Use the Design Check Set Dialog Box (page 1848), which is similar to the Label Set Dialog Box (page 1968), to create and edit design check sets.

**NOTE** Design checks validate the type of entity for which they were created, whether the entity is independent entity or a sub-entity in a tangent intersection group. For example, a curve design check validates a curve that is part of a spiral-curve-spiral group. However, a design check that validates the properties of a curve in spiral-curve-spiral groups does not validate the same properties of a standalone curve.

## Creating a Design Check Set

Use the Design Check Set dialog box to assign design checks to the individual alignment or profile elements that you want to validate.

Create design check sets from the Alignment ➤ Design Checks ➤ Design Check Sets and Profile ➤ Design Checks ➤ Design Check Sets collections in Toolspace on the Settings tab.

**To create a design check set**

1  In Toolspace, on the Settings tab, expand either the Alignment ➤ Design Checks or Profile ➤ Design Checks collection.

2  Right-click the Design Check Sets collection. Click New.

3  In the Design Check Set dialog box, on the Information tab (page 1849), enter a name and optional description of the design check set.

4  In the Design Check Set dialog box, on the Design Checks tab (page 1849), from the Type list, select a design check type.

5  From the <type> Checks list, select a design check.

6  Click Add.

   The design check is added to the design check set and appears in the table in the dialog box.

7  Repeat Steps 4 through 6 to add other design checks to the design check set.

8  If you are creating a profile design check set, use the Apply To column to specify the type of curve (sag, crest, or both) to apply each curve check.

9  Click OK.

**See also:**

- Alignment Design Checks (page 871)
- Applying Design Criteria to an Existing Alignment (page 1007)

### Quick Reference

Toolspace Shortcut Menu

   <drawing name> ➤ Alignment ➤ <right-click> Design Check Sets ➤ New

Dialog Box

   Design Check Set Dialog Box (page 1848)

## Editing a Design Check Set

Use the Design Check Set dialog box to edit existing design check sets.

Edit design check sets from the Alignment ➤ Design Checks ➤ Design Check Sets and Profile ➤ Design Checks ➤ Design Check Sets collections in Toolspace on the Settings tab.

**To edit a design check set**

1  In Toolspace, on the Settings tab, expand either the Alignment ➤ Design Checks or Profile ➤ Design Checks collection.

2  Right-click the design check set that you want to edit. Click Edit.

3  Use the Design Check Set dialog box, on the Design Checks tab (page 1849), to edit the design check set.

4  Click OK.

**See also:**

- [Alignment Design Checks](#) (page 871)

- [Applying Design Criteria to an Existing Alignment](#) (page 1007)

## Quick Reference

Toolspace Shortcut Menu (alignment design check sets)

    <drawing name> ➤ Alignment ➤ <right-click> Design Check Sets ➤ Edit

Toolspace Shortcut Menu (profile design check sets)

    <drawing name> ➤ Profile ➤ <right-click> Design Check Sets ➤ Edit

Dialog Box

    [Design Check Set Dialog Box](#) (page 1848)

# Superelevation

Use the Superelevation tab of the Alignment Properties dialog box to calculate and edit the superelevation specifications that you apply to roadway cross sections in the Corridor Model.

Methods of applying superelevation vary, depending on combinations of the Corridor Type and Cross Section Shape.

The parameters you set in the subassemblies, assemblies, and the corridor model determine how AutoCAD Civil applies these superelevation specifications.

# Superelevation on Undivided Roads

This section describes two situations for undivided roads and illustrates how superelevation is applied for each situation.

- The following illustration shows an undivided road as a corridor type and a crowned cross section shape. During superelevation, the simple crowned roadway undergoes adverse crown removal. In this illustration, the adverse crown removal is the distance from End Normal Crown to Reverse Crown:
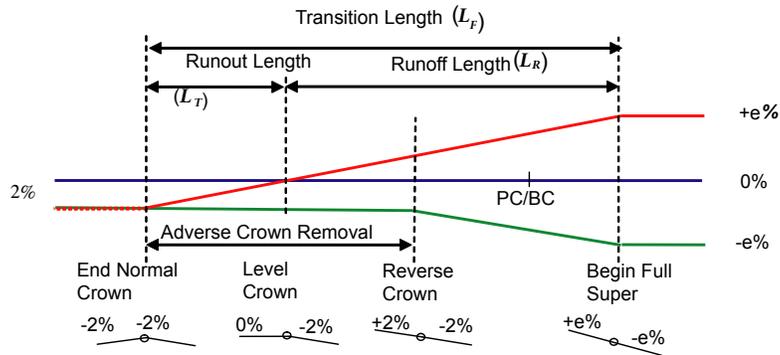
- The following illustration shows an undivided road as the corridor type and the cross section shape is planar. There is no crown, therefore no removal of adverse crown is necessary. This type of roadway typically has a downward slope in one direction on the unsuperelevated sections. Because of this, the distance required to achieve full superelevation varies depending on whether the superelevation continues in the direction of the downward slope or superelevates in the direction opposite to the normal downward slope. This example is typical of ramps, one-way traffic roads, and some service roads.
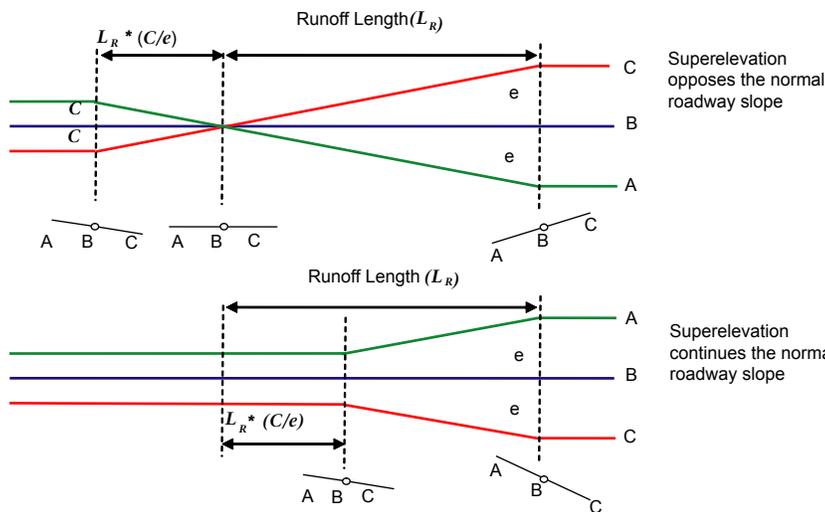


## Superelevation on Divided Roads

This section describes two situations for divided roads and illustrates how superelevation is applied for each situation.

- The following illustration shows a divided road as the corridor type and the cross section shape is planar. The roadway undergoes adverse crown removal during superelevation:
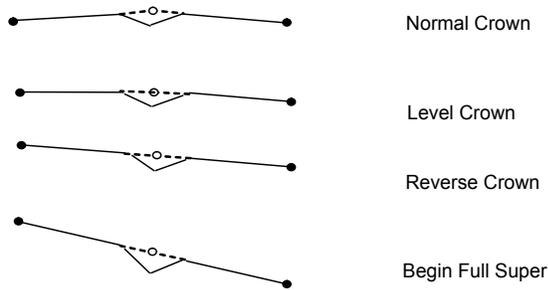
Normal Crown

Level Crown

Reverse Crown

Begin Full Super

■ The following illustration shows a divided road as the corridor type and a crowned cross section shape on each side. The roadway undergoes adverse crown removal during superelevation:



Normal Crown

Level Crown

Reverse Crown

Begin Full Super
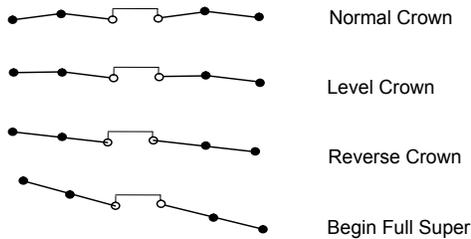
# Roadway Design Standards

Through the alignment properties, you can specify roadway design standards, including superelevation attainment method and minimum radius and transition length values.

Roadway design standards are contained in the design criteria file (page 2472). You can customize the design criteria file to reflect your local standards using the Design Criteria Editor Dialog Box (page 1843). For more information, see Design Criteria Files (page 867).

The XML-based design criteria file has three sections that you can use to calculate superelevation rates and superelevation transition stations on an alignment.

**Minimum Radius Tables**

A table of superelevation rates that you can apply to different types of roadways as a function of curve radius and design speed.

**Transition Length Tables**

A table of values you can use in the Superelevation Attainment Method formulas. You can use the table to calculate the distances between the critical superelevation transition points for different types of roadways as a function of curve radius and design speed. In many cases, the transition length tables provide the actual length of transition of superelevation runoff.

**Superelevation Attainment Methods**

Specifies how superelevation is applied, and the method that is used to calculate superelevation transition stations for different types of roadways. Each defined method specifies the formulas used to calculate the distances between the critical superelevation transition points. For more information, see Superelevation Specification Variables (page 880).

AutoCAD Civil supports two methods of superelevation attainment.

- Standard. Superelevation transition method that requires removal of adverse crown. This method is typically used on undivided, crowned roadways and divided roadways with crowned or planar sections.

- Planar. Superelevation transition method that does not involve removal of adverse crown. This method is typically used on undivided, planar-section roadways, such as ramps and service roads.

For more information and sample XML files, see Examples of Attainment Methods (page 881)

In the design criteria file, each attainment method, minimum radius table, and transition length table have unique, defined names. Use the Superelevation Specification Dialog Box (page 1776) to specify the name of the design criteria file.

# Superelevation Specification Variables

Use the following set of variables to calculate transition distances.

| | |
|---|---|
| {e} | The full superelevation rate. This rate is determined from the superelevation rate table, based on the design speed and curve radius. |
| {t} | The value that is read from the transition length tables, based on the design speed and the curve radius. This value may not be an actual length, but some other value, such as a transition rate from which the length can be calculated. |
| {c} | The unsuperelevated normal lane slope (positive). This value is defined by the user in the Superelevation Properties dialog box. |
| {s} | The unsuperelevated normal shoulder slope (positive). This value is defined by the user in the Superelevation Properties dialog box. |
| {w} | The nominal width from the pivot point to the outermost edge-of-traveled way. This value is defined by the user in the Superelevation Properties dialog box. |
| {l} | The length of the spiral, if a spiral is involved in the transition. This is the actual length of the spiral element in the curve group. |

The variables in the previous table are used to calculate the following distances:

| | |
|---|---|
| NCtoLC | Normal Crown point to Level Crown point (runout) |
| LCtoFS | Level Crown point to Full Super point (runoff) |
| LCtoRC | Level Crown point to Reverse Crown point |
| LCtoBC | Level Crown point to Beginning of Curve |
| NCtoFS | Normal Crown point to Full Super point (used instead of LCtoFS on undivided planar roads) for example the length of the superelevation transition |

| NCtoBC | Normal Crown point to Beginning of Curve (used instead of LCtoBC on undivided planar roads) |
| --- | --- |
| NStoNC | Normal Shoulder point to Normal Crown point (used for the Breakover Removal Method of superelevated shoulders) |

# Examples of Attainment Methods

Use this section to view examples of the XML formats you can use for various superelevation attainment methods.

**Standard Attainment Method Example**

The following example shows the XML format you can use to calculate transition stations for undivided, crowned roadways using the AASHTO standard methodology:

```
<SuperelevationAttainmentMethod name="AASHTO 2001 - Crowned Roadway">
<AttainmentStyle style="Standard"/>
<TransitionFormula type="LCtoFS" formula="{t}"/>
<TransitionFormula type="LCtoBC" formula ="2.0*{t}/3.0"/>
<TransitionFormula type="NCtoLC" formula ="{t}*{c}/{e}"/>
<TransitionFormula type="LCtoRC" formula ="{t}*{c}/{e}"/>
<TransitionFormula type="NStoNC" formula ="{t}*({s}-{c})/{e}">
<SuperelevationAttainmentMethod>
```

This example defines an attainment method whose name is "AASHTO 2001 - Crowned Roadway" which uses the standard adverse crown removal method of attaining superelevation. This example includes a calculation for the transition distance needed for shoulder breakover removal (type="NStoNC").

The following table describes the formulas:

| LCtoFS | The distance from the Level Crown Station to the Full Superelevation station is the value {t}, which is read from the selected transition length table. The formula assumes that the transition length table defines the runoff length. |
| --- | --- |
| | **NOTE** If there is a spiral, then LC to FS = spiral length. |
| LCtoBC | The distance from the Level Crown station to the Beginning of Curve station is 2/3 of the runoff length, {t}. |
| | **NOTE** If there is a spiral, then LC to BC = Lc to FS. |
| NCtoLC | The distance from Normal Crown station to the Level Crown Station (tangent runout) is calculated as the runoff length {t}, time the normal crown slope {c}, divided by the full superelevation rate {e}. The tangent runout length is extrapolated from the runoff length. |
| LCtoRC | The distance from the Level Crown station to the Reverse Crown station. (uses the same formula as NCtoLC) |
| NStoNC | Normal Shoulder point to Normal Crown point (used for Breakover Removal Method of superelevated shoulders) |

### Planar Transition Attainment Method Example

This example shows an undivided planar road The roadway is not crowned, and there is no adverse crown removal.

The overall transition distance can be different for left-turn and right-turn curves on a road where the right edge is on the low side because the road is tilted in one direction. The following illustration shows normalized slope superelevation, where the unsuperelevated road is tilted downward from left to right. Therefore the curve to the left requires a longer transition than the curve to the right:
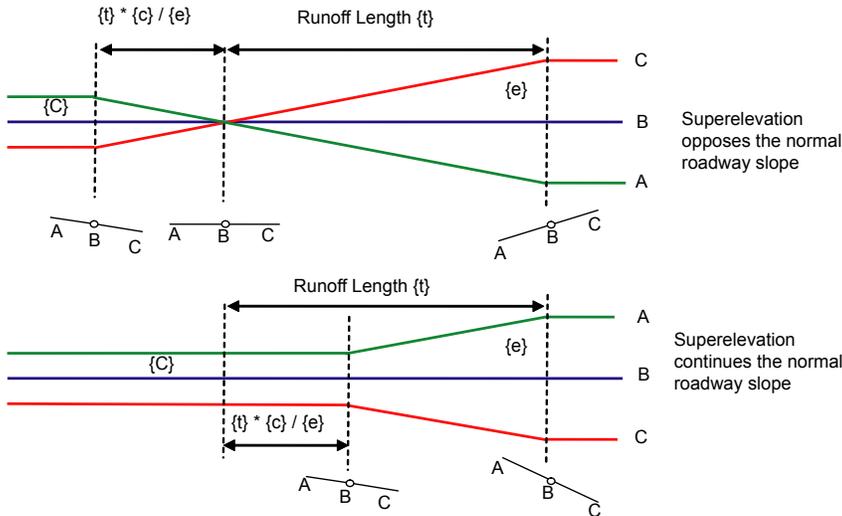


The Planar attainment method requires two formulas: one for curves that oppose the direction of the normal cross slope and one for curves that continue in the direction of the normal cross-slope. In the following example, the Continuing section defines the Normal Crown to Full Superelevation distance runoff length {t} (derived from the transition length tables), minus the runoff length times the normal roadway slope {c} divided by the full superelevation rate {e}. The second formula defines the distance from Normal Crown to the Beginning of Curve to be {t} times 2/3 minus {c} divided by {e}.

The Opposing section defines the overall transition distance to be the runoff length {t}. The distance to the Beginning of Curve is 2/3 {t}, and the distance between the Normal Crown and Level Crown stations is {t} * {c} / {e}

```
<SuperelevationAttainmentMethod name="Undivided Planar Roadway">
<TransitionStyle style="Planar"/>
<Continuing>
<TransitionFormula type="NCtoFS" formula="{t}-{t}*{c}/{e}"/>
<TransitionFormula type="NCtoBC" formula="{t}*(0.67-{c}/{e})"/>
</Continuing>
<Opposing>
<TransitionFormula type="LCtoFS" formula="{t}"/>
<TransitionFormula type="LCtoBC" formula="0.67*{t}"/>
<TransitionFormula type="NCtoLC" formula="{t}*{c}/{e}"/>
</Opposing>
<SuperelevationAttainmentMethod>
```

### Transitions Defined by Roadway Width and Transition Rate

Not all organizations use tables that give transition length directly. The following table defines the full

superelevation rate and the $\Delta$ as a function of design speed and curve radius. In this situation, the $\Delta$

value is used to derive the transition length based on the nominal width of the roadway. The Transition Length tables define the $\Delta$ value instead of the actual transition length.

| Radius (m) | 90 km/h | | 100km/h | | 110km/h | | 120km/h | |
|---|---|---|---|---|---|---|---|---|
| | E% | $\Delta$ | E% | $\Delta$ | E% | $\Delta$ | E% | n/a |
| 7000 | NC | n/a | NC | n/a | NC | n/a | NC | n/a |
| 5000 | NC | n/a | NC | n/a | NC | n/a | 2.0 | 0.31 |
| 3000 | 2.0 | 0.39 | 2.0 | 0.34 | 2.0 | 0.32 | 2.0 | 0.31 |
| 2500 | 2.0 | 0.39 | 2.0 | 0.34 | 2.0 | 0.32 | 2.0 | 0.31 |
| 2000 | 2.0 | 0.39 | 2.0 | 0.34 | 2.0 | 0.32 | 2.3 | 0.32 |
| 1500 | 2.0 | 0.39 | 2.0 | 0.34 | 2.2 | 0.33 | 3.0 | 0.33 |
| 1400 | 2.0 | 0.39 | 2.0 | 0.34 | 2.4 | 0.33 | 3.2 | 0.34 |
| 1300 | 2.0 | 0.39 | 2.0 | 0.34 | 2.6 | 0.33 | 3.5 | 0.34 |
| 1200 | 2.0 | 0.39 | 2.2 | 0.35 | 2.8 | 0.34 | 3.8 | 0.35 |
| 1000 | 2.0 | 0.39 | 2.6 | 0.36 | 3.7 | 0.35 | 4.5 | 0.37 |
| 900 | 2.2 | 0.40 | 2.9 | 0.37 | 3.7 | 0.36 | 5.0 | 0.38 |
| 800 | 2.5 | 0.40 | 3.3 | 0.38 | 4.2 | 0.38 | 5.7 | 0.39 |
| 700 | 2.9 | 0.41 | 3.7 | 0.39 | 4.8 | 0.39 | 6.0 | 0.40 |
| 600 | 3.4 | 0.42 | 4.4 | 0.41 | 5.6 | 0.41 | | |
| 500 | 4.0 | 0.44 | 5.2 | 0.43 | 6.0 | 0.42 | | |
| 400 | 5.0 | 0.46 | 6.0 | 0.45 | | | | |
| 300 | 6.0 | 0.48 | | | | | | |

The following example shows attainment methods and formulas for two types of roadways based on the previous table. The variable {w} is the nominal roadway width from pivot point to edge-of traveled-way, defined in the Superelevation Properties dialog box.

```
<SuperelevationAttainmentMethod name="Unspiraled ramp">
<TransitionStyle style="Planar"/>
<Continuing>
<TransitionFormula type="NCtoFS" formula="100*{e}*{w}/{t}"/>
<TransitionFormula type="NCtoBC" formula="66.7*{e}*{w}/{t}"/>
</Continuing>
<Opposing>
<TransitionFormula type="LCtoFS" formula="100*{e}*{w}/{t}"/>
<TransitionFormula type="LCtoBC" formula="66.7*{e}*{w}/{t}"/>
<TransitionFormula type="NCtoLC" formula="100*{c}*{w}/{t}"/>
<TransitionFormula type="LCtoRC" formula="100*{c}*{w}/{t}"/>
</Opposing>
</SuperelevationAttainmentMethod>
<SuperelevationAttainmentMethod name="Unspiraled 2 way roadway">
<TransitionStyle style="Standard"/>
<TransitionFormula type="LCtoFS" formula="100*{e}*{w}/{t}"/>
<TransitionFormula type="LCtoBC" formula="66.7*{e}*{w}/{t}"/>
<TransitionFormula type="NCtoLC" formula="100*{c}*{w}/{t}"/>
<TransitionFormula type="LCtoRC" formula="100*{c}*{w}/{t}"/>
<SuperelevationAttainmentMethod>
```

# Adding Superelevation Specifications

Superelevation specifications define travel lane and shoulder slopes that you can use with subassemblies when you create corridor models.

The Superelevation Specification Dialog Box (page 1776) lists each alignment curve group as a Superelevation Region. Enter superelevation data for key points on an alignment curve geometry and assign the design criteria file in this dialog box. After you calculate the superelevation, you can edit these specifications on the Superelevation tab.

**WARNING** If the horizontal geometry violates minimum standards in the look-up tables, for example, if the radius of the curve is too small for the design speed, the Event Viewer posts a warning. It is highly recommended that you set the Show Event Viewer property to Yes in the General section of the Edit Feature Settings - Alignment Dialog Box (page 1767).

**To calculate superelevation values**

1   Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties .

2   In the Alignment Properties dialog box, click the Superelevation (page 1756) tab.

3   Click  Set Superelevation Properties to open the Superelevation Specification Dialog Box (page 1776). Each curve on the horizontal alignment is listed as a Superelevation Region.

    **NOTE** Before you enter the Superelevation Specifications, you must set the appropriate design speeds on the Design Criteria tab (page 1754) in the Alignment Properties dialog box.

4   In the Design Rules section, click the Design Criteria File Name cell and browse to a design criteria file. Depending on the file you select, you can edit the Superelevation Rate Table, the Transition Length Table, and the Attainment Method.

    **NOTE** You can edit the roadway design standards to reflect your local standards. For more information, see Editing a Design Criteria File (page 870).

**5** Click OK. If there is previous superelevation data, you receive a warning that existing data will be overwritten. The Superelevation tab is populated with descriptions and values for each region.

**6** Click in a cell to make edits. Optionally, click  Add A Transition Station to add transition stations or click  Delete Transition Station to remove a transition station.

**See also:**

■ Superelevation Specification Dialog Box (page 1776)

### Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties

Toolspace Shortcut Menu

Prospector tab: Right-click <alignment item> ➤ Properties

Object Shortcut Menu

Right-click Alignment object ➤ Alignment Properties ➤ Superelevation tab

Command Line

EditAlignmentProperties

Dialog Box

Superelevation Tab (Alignment Properties Dialog Box) (page 1756).

# Alignment Styles

You can use Alignment Styles to control the visual display of each alignment component.

Changes made to a style are applied to all alignments that use the style.

Use styles to control the appearance of the alignment object components, as well as the types and appearance of alignment labels.

You can create specific styles to use for the different phases of a project. For example, you can create one style to use in the design layout phase and another style to use for plotting.

Access alignment styles using these methods:

| If you want to access alignment styles... | Then... |
| --- | --- |
| In the drawing | Right-click an alignment ➤ Edit Alignment Style. |
| On the Settings tab in Toolspace | Right-click a style in the Alignment Styles folder ➤ Edit. |

**NOTE** When you click New in a collection folder, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a style from an existing style, right-click the style. Click Copy and save the style with a new name.

You use the tabs in the Alignment Style dialog box to set style information:

■ Information. Specifies name, description, and creation information.

■ Design. Specifies certain grip edit behavior.

■ Markers. Specifies marker styles and marker placement options for alignment points.

■ Display. Lists the alignment components and display options as well as the choice of 2D or 3D visibility. You can display components in a 2D plan view or a 3D view using other tools, such as Orbit or Vpoint.

■ Summary. Lists all the style properties.

# Applying an Alignment Style

You can apply different styles to alignments to change the visual display of the alignment components.

You can create specific styles to use for the different phases of a project. For example, you can create one style to use in the design layout phase and another style to use for plotting.

**To apply an alignment style to an existing alignment**

1 In the drawing, click an alignment.

2 Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties .

3 In the Alignment Properties (page 863) dialog box, on the Information tab, use the Object Style list to select a style.

4 Click OK.

## Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties 

Toolspace Shortcut Menu

Prospector tab: Right-click alignment object ➤ Properties

Object Shortcut Menu

Right-click <alignment object> ➤ Alignment Properties

Dialog Box

Alignment Properties (page 863)

# Copying or Editing Alignment Styles

Copy or edit an alignment style by clicking an existing style, modifying it, and saving it with a new name.

**To copy or edit an alignment style**

1 In Toolspace, on the Settings tab, expand the Alignment Styles collection and right-click an existing style.

**2** Click Copy or Edit.

**3** In the Alignment Style dialog box, click the Information (page 1759) tab. Enter a new name and description for the alignment style.

**4** Click the Design (page 1759) tab. Specify the radius snap and the snap values.

**5** Click the Markers (page 1759) tab. Specify the display properties for the alignment.

**6** Click the Display (page 1760) tab. Specify the display properties for the alignment.

**7** To view a summary of information about the style, click the Summary (page 1760) tab.

**8** Click Apply.

### Quick Reference

Ribbon

Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Edit Alignment Style

Toolspace Shortcut Menu

Settings tab: Right-click alignment style item ➤ Copy

Object Shortcut Menu

Right-click <alignment object> ➤ Edit Alignment Style

Dialog Box

Alignment Style Dialog Box (page 1758)

# Creating Alignments

Create alignments in many ways using AutoCAD Civil, such as creating them from polylines, and using the Alignment Layout Tools.

When you create an alignment, you can use the criteria-based design (page 2471) feature to ensure that your alignment design meets minimum local standards. For more information, see Criteria-Based Alignment Design (page 866).

**NOTE** You can also create alignments either from pipe networks or by importing them from LandXML files. For more information, see Creating Alignments from Pipe Network Parts (page 1196) and LandXML Import and Export (page 1673).

# Creating an Alignment with the Alignment Layout Tools

Create an alignment with the Alignment Layout Tools.

Use the layout command to create an alignment. Specify a unique name, site (if desired), style and label set, and criteria-based design (page 2471) options. Then, use the Alignment Layout Tools to draw the alignment.

This toolbar provides two types of alignment creation tools:

■ Freehand drawing tools for lines, curves, and spirals.

■ Constraint-based tools that define individual entities by parameters such as length and radius.
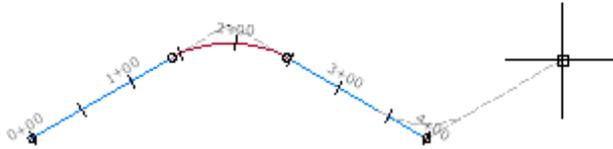
**To create an alignment using the alignment layout tools**

1  Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools .

2  In the Create Alignment - Layout Dialog Box (page 1763), enter a unique name for the alignment.

3  Enter an optional description.

4  Enter a starting station value.

5  On the General tab, specify the following settings:

■ Site
Specify a site with which to associate the alignment or accept the default <None>. For more information, see Understanding Sites (page 719).

■ Alignment Style

■ Alignment Layer

■ Alignment Label Set

6  On the Design Criteria tab, specify a Starting Design Speed to apply to the alignment starting station.

This design speed is applied to the entire alignment. Additional design speeds can be applied at other stations after the alignment has been created. For more information, see Alignment Properties (page 863).

**NOTE** If you do not want to apply design criteria to the alignment, proceed to Step 8.

7  To associate design standards with the alignment, select the Use Criteria-Based Design check box.

For more information, see Criteria-Based Alignment Design (page 866).

On the Design Criteria tab, specify the following settings:

■ Use Design Criteria File
Select this check box to associate a design criteria file with the alignment. The default design criteria file location and the Default Criteria properties becomes available when you select the check box.

**NOTE** The first design criteria file found in the *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>* directory is used by default when the criteria-based design feature is used.

■ Use Design Check Set
Select this check box to associate a design check set with the alignment. The design check set list becomes available when you select the check box. Select a design check set from the list.

**8** Click OK to display the Alignment Layout Tools toolbar.

**9** Use the commands on the Alignment Layout Tools toolbar to draw the alignment. For more information, see Alignment Layout Tools (page 1747) and Adding Lines, Curves, and Spirals to an Alignment (page 898).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools

Menu

Alignments menu ➤ Create Alignment By Layout

Command Line

CreateAlignmentLayout

Dialog Box

Create Alignment - Layout Dialog Box (page 1763)

# Drawing Tangent-Tangent Alignments

Use the Alignment Layout Tools to create and edit alignments.

You can use either of the following commands to create a quick layout of the alignment. The lines and curves you create are fixed entities that can be edited without losing tangency.

**Tangent-Tangent No Curves**

Use this command to create a quick layout of an alignment. Specify tangent points of intersection (PI). Optionally add curves later.
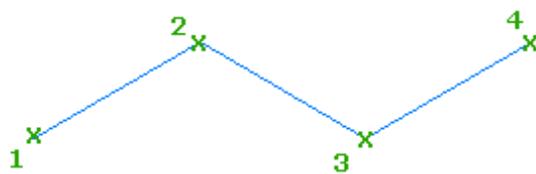


**Tangent-Tangent With Curves**

Use this command to lay out an alignment, automatically creating curves at points of intersection (PI). Curve parameters are the current values in the Curve and Spiral Settings dialog box.

---

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

**See also:**

■ Curve and Spiral Settings (page 861)

**To draw an alignment using the tangent-tangent method with no curves**

1   Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools .

2   In the Create Alignment-Layout (page 1763) dialog box, enter creation information.

3   On the Alignment Layout Tools toolbar, click  Tangent-Tangent (No Curves).

4   Specify the start point.

5   Continue to specify points. Press Enter to end the command, or click another command from the Alignment Layout Tools toolbar.

**To draw an alignment using the tangent-tangent method with curves**

1   Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools .

2   In the Create Alignment-Layout (page 1763) dialog box, enter creation information.

3   On the Alignment Layout Tools toolbar, click  Tangent-Tangent (With Curves).

4   Specify the start point.

5   Continue to specify points. Press Enter to end the command or select another command from the Alignment Layout Tools.

   Curves are created to comply with the values in one of the following places:

   ■ If the alignment has design criteria (page 866) applied to it, the minimum radius for the current design speed is applied to the curve. If no solution can be found with the minimum radius value, the curve is not drawn.

   ■ If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is applied to the curve. For more information, see Curve and Spiral Settings (page 861).

**To insert a PI on a fixed line**

**1** Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏.

**2** On the Alignment Layout Tools toolbar, click △ Insert PI.

**3** In the drawing, pick a point near a tangent, where you want to insert the PI.

**To remove a PI and combine two tangents**

**1** Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏.

**2** On the Alignment Layout Tools toolbar, click △✗ Delete PI.

**3** In the drawing, pick a point near the PI you want to delete.

**To break apart the PI of an incoming or outgoing tangent endpoint**

**1** Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏.

**2** On the Alignment Layout Tools toolbar, click |△\ Break-Apart PI.

**3** In the drawing, pick a point near the PI you want to break apart.

**4** Specify the distance to split the PI by picking two points or entering a value on the command line. This value is the distance each line is shortened from the PI.

**5** Specify the second point.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Alignment Creation Tools ⬚
Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏

Menu

Click Alignments menu ➤ Create Alignment By Layout
Click Alignments menu ➤ Edit Alignment Geometry

Command Line

CreateAlignmentLayout
EditAlignment

Dialog Box

Create Alignment - Layout Dialog Box (page 1763)
Alignment Layout Tools (page 1747)

# Creating an Alignment from Graphic Entities

Create an alignment from selected lines, curves, or polylines.

This command converts a series of graphic entities to an alignment, and optionally adds curves between tangents.
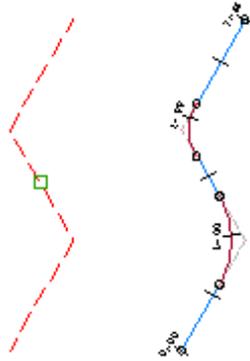
**To create an alignment from graphic entities**

1 Draw one or more lines, curves, or polylines in the drawing.

2 Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Objects ⬚.

3 Select the graphic entities.

   **NOTE** Enter XREF at the command line to select entities from within the xref drawing. See AutoCAD Help for more information about external drawing references.

4 In the Create Alignment From Objects Dialog Box (page 1760), enter a unique name for the alignment.

5 Enter an optional description.

6 Enter a starting station value.

7 On the General tab, specify the following settings:
   - Site
     Specify a site with which to associate the alignment or accept the default <None>. For more information, see Understanding Sites (page 719).

   - Alignment Style

   - Alignment Layer

   - Alignment Label Set

8 On the Design Criteria tab, specify a Starting Design Speed to apply to the alignment starting station.

   This design speed is applied to the entire alignment. Additional design speeds can be applied at other stations after the alignment has been created. For more information, see Alignment Properties (page 863).

   **NOTE** If you do not want to apply design criteria to the alignment, proceed to Step 8.

9 To associate design standards with the alignment, select the Use Criteria-Based Design check box.

   For more information, see Criteria-Based Alignment Design (page 866).

   On the Design Criteria tab, specify the following settings:
   - Use Design Criteria File
     Select this check box to associate a design criteria file with the alignment. The default design criteria file location and the Default Criteria properties become available when you select the check box.

> **NOTE** The first design criteria file found in the *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>* directory is used by default when the criteria-based design feature is used.

- Use Design Check Set
  Select this check box to associate a design check set with the alignment. The design check set list becomes available when you select the check box. Select a design check from the list.

**10** Click OK.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Objects ⬚

Menu

Click Alignments menu ➤ Create Alignment From Polyline

Command Line

CreateAlignmentEntities

Dialog Box

Create Alignment From Objects Dialog Box (page 1760)

# Creating an Alignment Reference

Create a reference to an existing alignment in a data shortcut (page 2472) or a Vault project

The alignment reference is a lightweight read-only copy of the original, but it gives you access to alignment data for other objects. Before you create the alignment reference, it must exist in the current project collection on the Toolspace Prospector tab

**See also:**

- Creating Data Shortcuts (page 131)
- Using Vault (page 138)

**To create a reference to a data shortcut alignment**

**1** In Toolspace, on the Prospector tab, ensure that the correct working folder is identified on the Data Shortcuts node.

**2** Expand the Data Shortcuts ➤ Alignments collection.

**3** Right-click the desired alignment, and then click Create Reference.

The Create Alignment Reference dialog box is displayed, in which you can optionally change the properties of the reference alignment, as described in the following steps.

**4** In the Create Alignment Reference dialog box, change the source of the reference by selecting a source in the Source Alignment list.

**5** Specify a Site with which to associate the alignment or accept the default <None>. For more information, see Understanding Sites (page 719).

**6** Enter a Name for the alignment.

> **NOTE** To name the alignment, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**7** Specify an Alignment Style or accept the default style.

For more information, see Alignment Styles (page 885).

**8** Click ⬚ to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the alignment is placed on the current layer.

**9** Specify a Alignment Label Set or accept the default label set.

**10** Click OK to create the alignment reference.

The alignment name is displayed under the Alignments collection in the Prospector tree with a ⬚ next to it.

**To create a reference to an alignment in a Vault project**

**1** In Toolspace, on the Prospector tab, in Master View, expand the Projects <project name> ➤ Alignments collection.

**2** Right-click the desired alignment, and then click Create Reference.

The Create Alignment Reference dialog box is displayed, in which you can optionally change the properties of the reference alignment, as described in the following steps.

**3** In the Create Alignment Reference dialog box, change the source of the reference by selecting a source in the Source Alignment list.

**4** Specify a Site with which to associate the alignment or accept the default <None>. For more information, see Understanding Sites (page 719).

**5** Enter a Name for the alignment.

> **NOTE** To name the alignment, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**6** Specify an Alignment Style or accept the default style.

For more information, see Alignment Styles (page 885).

**7** Click ⬚ to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the alignment is placed on the current layer.

**8** Specify an Alignment Label Set or accept the default label set.

**9** Click OK to create the alignment reference.

The alignment name is displayed under the Alignments collection in the Prospector tree with a ⬚ next to it.

**Quick Reference**

Toolspace Right-Click Menu (for data shortcut project)

Prospector tab: Data Shortcuts ➤ Alignments ➤ <alignment name> ➤ Create Reference

Toolspace Right-Click Menu (for Vault project)

Prospector tab: Projects ➤ <project name> ➤ Alignments ➤ <alignment name> ➤ Create Reference

Command Line

CreateAlignmentReference

Dialog Box

Create Alignment Reference (page 2199)

# Creating Offset Alignments

Create parallel alignments that are dynamically linked to an existing alignment.

In a single operation, create multiple offset alignments on one or both sides of the parent alignment. The offset distance is specified separately for left and right sides.



Left and right sides are determined from a position facing toward the end of the parent alignment.

The offset alignments retain their offset distance if the parent alignment is edited. The only geometric attribute of the offset alignments to be edited directly is offset distance.

**To create offset alignments**

1   Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Offset Alignment .

2   In the Create Offset Alignments (page 1765) dialog box, select the following attributes:
   ■ Parent alignment
   ■ Name template
   ■ Station range for the offset
   ■ Number of offsets on each side
   ■ Incremental offset distance on each side
   ■ Design Criteria (optional): design speed, criteria set

3   On the General tab, enter the site, alignment style, layer, and label set for the offset alignments.

**4** If you want to apply criteria-based design to the offset alignments, click the Design Criteria tab. Enter the required design speed. Click Use Criteria-Based Design, and then select the criteria for the design.

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Offset Alignment

Command Line
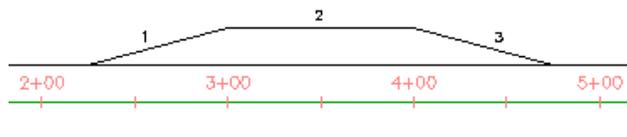
CreateOffsetAlignment

Dialog Box

Create Offset Alignments Dialog Box (page 1765)

# Widenings

You can apply widening for a specified length along any type of alignment.



Each widening (page 2487) includes transition regions at the entry and the exit. In the figure, region 1 is the *entry transition*, and region 3 is the *exit transition*, based on the direction of the centerline alignment. Default values for the length and shape of these transitions are determined by command settings for the AddWidening command.

The transitions extend forward and back from the specified widening start and end stations. In the example shown, the widening starts at station 3+00 and the transition length is 75, so the entry transition starts at 2+25.

Curb return alignments in intersections can be edited to add widenings at one or both ends. This type of widening typically forms a turn lane at the entry to a curb return or a merge lane at the exit from a curb return.
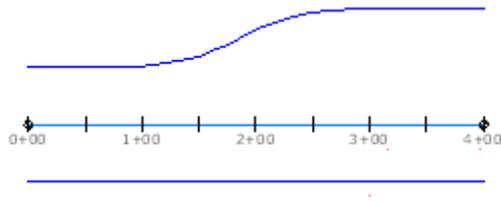
**See also:**

■ Editing Offset Alignments and Widenings (page 1005)

■ Editing Offsets in Intersections (page 1445)

# Creating Widening

Create a widened region along an existing alignment.

Use this command to create features such as turn lanes, bus bays, and roadside parking areas.

You can add a widening to any alignment type, or to another widening. In the creation process, you specify the following attributes:

■ Whether the widening is a separate alignment, or part of the parent alignment

■ The start and end stations along the parent alignment

■ The offset width

■ The side (left or right) to widen, based on facing in the forward direction along the alignment

A widening can be part of the parent alignment only if the parent is a dynamic offset or curb return.

**See also:**

■ Editing Offset Alignments and Widenings (page 1005)

**To create a widening**

1 Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Widening .

2 Click an alignment to attach the widening to. This is also known as the parent alignment.

3 Specify whether to create the widening as a new alignment, or part of the parent alignment.

4 Specify start and end stations for the widening, either by clicking on the parent alignment or entering numeric station values.

5 Specify the offset width.

6 Specify whether the widening is offset to the left or right of the parent alignment. The widening is created, and its default parameters are displayed in the Offset Parameters window.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Widening

Command Line

AddWidening

# Adding Lines, Curves, and Spirals to an Alignment

Use the constraint-based commands on the Alignment Layout Tools toolbar, to add a fixed entity (page 2474), a free entity (page 2474), or a floating entity (page 2474) (lines, curves, spiral-curve-spiral groups and spirals) to the alignment.

The Alignment Layout Tools toolbar provides you with an infinite number of ways to solve various problems by using different constraints.

Using the constraint-based commands, you can fit an alignment by working inward from two known points at the beginning and end of the design area. You can also create alignment entities in sequential order.

With the established tangency rules, you can edit the parameters of any alignment entity in real-time and still maintain tangency.

# Entity Descriptions

This section contains descriptions of fixed, free, and floating alignment entities.

**Fixed Entity**

A fixed entity has the following characteristics:

■ It is defined by specifying parameters, such as points or radius.

■ Its parameters are fixed in their position. Only you can change the parameters that define this entity.

■ It is not dependent on any other entity to define its geometry or to maintain tangency, but its length can be affected by another entity that is attached to it.

■ It can be initially created with a tangency association. Fixed entities that start from the beginning of another fixed entity can be initially tangent to the other entity, but the tangency is not maintained when either entity is edited.

**Floating Entity**

A floating entity has the following characteristics:

■ It is defined by the parameters that you specify and is always tangent to one entity.

■ It is always tangent to the entity it is attached to. A floating entity cannot be attached to a free entity, but a free entity can be attached to a floating entity.

■ It can be attached only to another floating entity or a fixed entity.

■ It is dependent on being tangent to one other entity to define its geometry.

**Free Entity**

A free entity has the following characteristics:

■ It is defined by the parameters that you specify and is always tangent to an entity before and an entity after.

■ It must have at least two other entities to add to.

■ It is dependent on two other entities to define its geometry.

■ It can only be attached between either two fixed entities, two floating entities, or one fixed and one floating entity, but in any order.

# Adding Lines

Add fixed, free, or floating lines to build constraint-based alignment geometry for the areas of your design that require maintaining tangency.

In most cases the length of the line automatically adjusts to meet the tangency point of the next or previous entity.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

## Adding Fixed Lines

Add fixed lines to build constraint-based alignment geometry for the areas of your design that require maintaining tangency.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

## Adding a Fixed Line Between Two Points

Add a fixed line between two points.

The entity position is not affected by edits to adjacent entities.
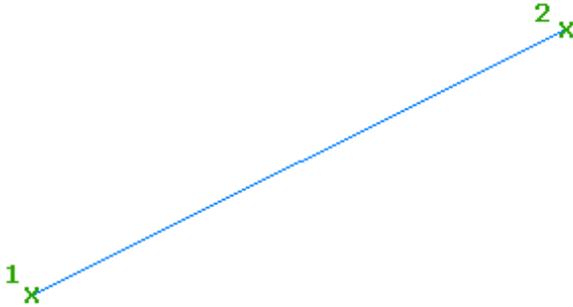
**To add a fixed line (two points)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✐ .

2 On the Alignment Layout Tools toolbar, click ✐ Fixed Line (Two Points).

3 Specify the start point.

4 Specify the end point. A line preview is displayed.

**TIP** Edit the line direction or location by moving either point or move the line using the middle grip.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏️

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

✏️ Fixed Line (Two Points)

## Adding a Fixed Line with a Specified Length to a Curve End

Add a fixed line of a specified length to the end of an existing curve.

The direction of the line matches the direction of the attachment curve (1). The specified line length (2) is affected only if a floating or free entity is attached to it.
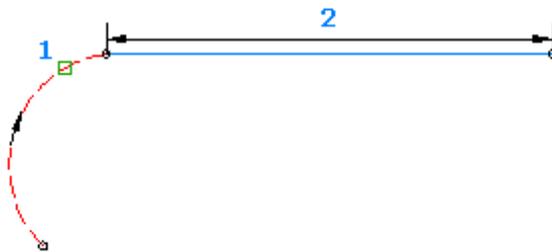


The line is tangent to the end of the selected fixed or floating entity when it is created. If you edit the line or the entity it is attached to, tangency is not maintained.

**To add a fixed line (from curve end, length)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏️ .

2   On the Alignment Layout Tools toolbar, click ✏️ Fixed Line (From Curve End, Length).

3   Select an entity end for the start point and direction.

4   Specify the length. A line preview is displayed.

By specifying the start point at the end of an entity, a direction, and length, you create a fixed two-point line alignment entity. The result is a line through two points.

**TIP** You can edit either of the two points or the middle grip, but the initial tangency will not be maintained.

**Quick Reference**

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

Fixed Line (From Curve End, Length)

## Adding a Fixed Line by Best Fit

Add the most probable fixed line through a series of using a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

Regression points can be added, removed, or modified after the entity has been created.
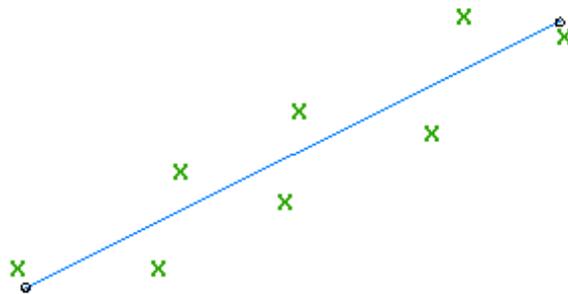


**See also:**

■ Editing Best Fit Alignment Entities (page 1003)

**To create a fixed line by best fit from AutoCAD Civil points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click Fixed Line - Best Fit.

3 In the Line By Best Fit Dialog Box (page 1989), select From COGO Points.

4 Select two or more AutoCAD Civil points. Enter G to select a point group or N to enter points by number.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

5 Press Enter to complete the command.

6 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the line:

- Click ![icon] to create the line and keep the Regression Data vista open.

- Click ![icon] to create the line and close the Regression Data vista.

**To create a fixed line by best fit from AutoCAD points**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon] .

**2** On the Alignment Layout Tools toolbar, click ![icon] Fixed Line - Best Fit.

**3** In the Line By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

**4** Select two or more AutoCAD points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the line:

- Click ![icon] to create the line and keep the Regression Data vista open.

- Click ![icon] to create the line and close the Regression Data vista.

**To create a fixed line by best fit from existing entities**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon] .

**2** On the Alignment Layout Tools toolbar, click ![icon] Fixed Line - Best Fit.

**3** In the Line By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**4** Select one or more of the entities listed on the command line.

---

**NOTE** You can select several types of entities listed on the command line.

---

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

**5** If you selected a profile object, specify the starting and ending station in the Specify Station Range Dialog Box (page 1992).

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the line:

- ■ Click ![save icon] to create the line and keep the Regression Data vista open.

- ■ Click ![checkmark icon] to create the line and close the Regression Data vista.

**To create a fixed line by best fit by clicking on screen**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon].

**2** On the Alignment Layout Tools toolbar, click ![icon] Fixed Line - Best Fit.

**3** In the Line By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**4** Select a starting point and at least one other point.

> **NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the line:

- ■ Click ![save icon] to create the line and keep the Regression Data vista open.

- ■ Click ![checkmark icon] to create the line and close the Regression Data vista.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Click Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

![icon] Fixed Line - Best Fit

![icon] Edit Best Fit Data For All Entities

Dialog Box

Alignment Layout Tools (page 1747)
Line By Best Fit Dialog Box (page 1989)
Regression Data Vista (page 1990)

## Adding Floating Lines

Add floating lines to build constraint-based alignment geometry for the areas of your design that require maintaining tangency during edits.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

## Adding a Floating Line through a Point from a Curve

Add a floating line from a point on an existing curve to a specified point.

The direction of the attachment curve (1) and the location of the pass-through point (2) determine the attachment point.

**To add a floating line (from curve, through point)**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2  On the Alignment Layout Tools toolbar, click ⌒ Floating Line (From Curve, Through Point).

3  Select the end of the curve to which you want to add the floating line.

4  Specify the end point.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

## Adding a Floating Line of a Specified Length to a Curve End

Adds a floating line, with a specified length, to the end of a curve.

The direction of the attachment curve (1) determines the line direction.
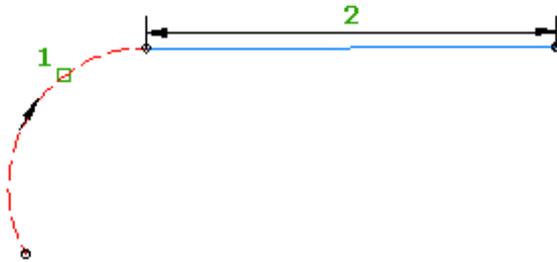


This type of entity always starts at the end of the entity it is attached to. The line is always tangent to the end of a fixed or floating curve. Unlike an entity where the location is defined by a pass-through point, this type of entity is not pinned to a location in the drawing. Therefore, the entity geometry moves with the entity to which it is attached.

**To add a floating line (from curve end, length)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⟋ Floating Line (From Curve End, Length).

3   Select the end of the curve to which you want to add the floating line.

4   Specify a length by picking two points in the drawing, or by entering a length value on the command line.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

⟋ Floating Line (From Curve End, Length)

## Adding a Floating Line by Best Fit

Adds the most probable floating line from a point on an existing entity through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

The direction of the attachment curve (1) and the path through the selected regression points (2) determines the location of the attachment point.
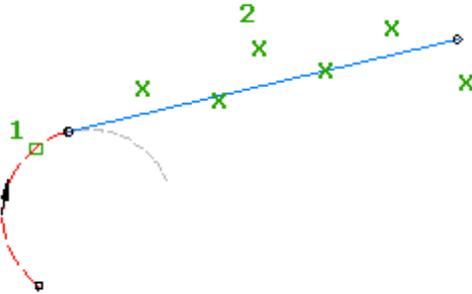


**See also:**

■ Editing Best Fit Alignment Entities (page 1003)

**To create a floating line by best fit using AutoCAD Civil points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2 On the Alignment Layout Tools toolbar, click 📏 Floating Line - Best Fit.

3 Select a curve entity for the start point and direction.

 If you select the beginning of the curve entity, the best fit line will precede the curve entity. If you select the end of the curve entity, the best fit line will succeed the curve entity.

4 In the Line By Best Fit Dialog Box (page 1989), select From COGO Points.

5 Select one or more AutoCAD Civil points. Enter G to select a point group or N to enter points by number.

 As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

6 Press Enter to complete the command.

7 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

 As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

8 Create the line:

 ■ Click 💾 to create the line and keep the Regression Data vista open.

 ■ Click ✔ to create the line and close the Regression Data vista.

**To create a floating line by best fit from AutoCAD points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click  Floating Line - Best Fit.

3 Select a curve entity for the start point and direction.

If you select the beginning of the curve entity, the best fit line will precede the curve entity. If you select the end of the curve entity, the best fit line will succeed the curve entity.

4 In the Line By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

5 Select one or more AutoCAD points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

6 Press Enter to complete the command.

7 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

8 Create the line:

■ Click  to create the line and keep the Regression Data vista open.

■ Click  to create the line and close the Regression Data vista.

**To create a floating line by best fit from existing entities**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click  Floating Line - Best Fit.

3 Select a curve entity for the start point and direction.

If you select the beginning of the curve entity, the best fit line will precede the curve entity. If you select the end of the curve entity, the best fit line will succeed the curve entity.

4 In the Line By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

5 Select one or more of the entities listed on the command line.

**NOTE** You can select several types of entities listed at the command line.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

6 If you selected a profile object, specify the starting and ending station in the Specify Station Range Dialog Box (page 1992).

7 Press Enter to complete the command.

8 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the line:

- ■ Click ![save icon] to create the line and keep the Regression Data vista open.

- ■ Click ![check icon] to create the line and close the Regression Data vista.

**To create a floating line by best fit by clicking on screen**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon] .

2 On the Alignment Layout Tools toolbar, click ![icon] Floating Line - Best Fit.

3 Select a curve entity for the start point and direction.

   If you select the beginning of the curve entity, the best fit line will precede the curve entity. If you select the end of the curve entity, the best fit line will succeed the curve entity.

4 In the Line By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

5 Select at least one other point.

   **NOTE** You can use OSNAP or transparent commands to select points.

   As you select points in the drawing window, an X marks each regression point and a temporary, dashed line is displayed in real time.

6 Press Enter to complete the command.

7 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

   As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

8 Create the line:

- ■ Click ![save icon] to create the line and keep the Regression Data vista open.

- ■ Click ![check icon] to create the line and close the Regression Data vista.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

![icon] Floating Line - Best Fit

![icon] Edit Best Fit Data For All Entities

Dialog Box

## Adding Free Lines

Add free lines to build constraint-based alignment geometry for the areas of your design that require maintaining tangency between two curves.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

## Adding a Free Line Between Two Curves

Add a free line between two existing curves.

This entity maintains tangency to both attachment curves (2). There are four possible solutions for this entity, depending on the direction of the curves.
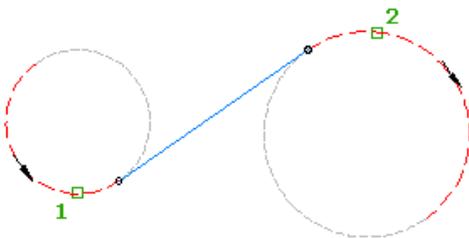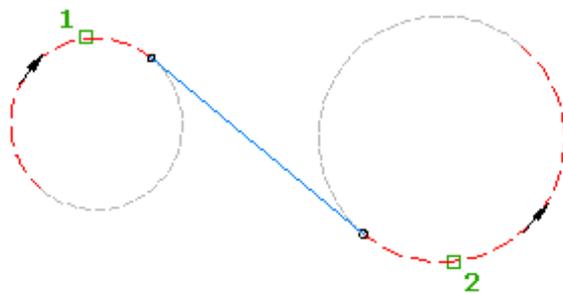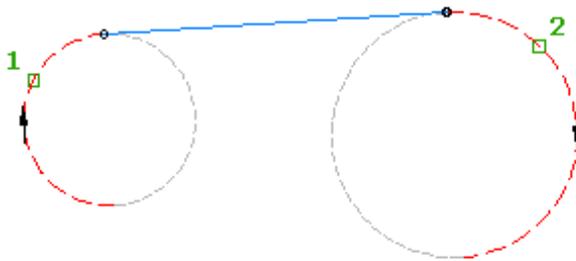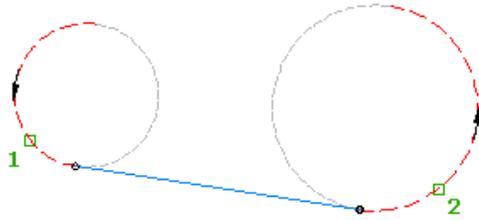
**To add a free line (between two curves)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2 On the Alignment Layout Tools toolbar, click 📐 Free Line (Between Two Curves).

3 Specify the curve entity from which you want to add the free line.

4 Specify the curve entity to which you want to add the free line.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

📐 Free Line (Between Two Curves)

# Adding Curves

Add fixed, free, or floating curves to build constraint-based alignment geometry for the areas of your design that require maintaining tangency.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

## Adding Fixed Curves

Add a fixed curve through three specified pass-through points.

These commands are similar to an AutoCAD arc, but have a third point along the arc.

## Adding a Fixed Curve with Three Points

Add a fixed three-point curve by specifying three points.

The entity position is not affected by edits to adjacent entities.



**To add a fixed curve (with three points)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2 On the Alignment Layout Tools toolbar, click 🌰 Fixed Curve (Three Point).

3 Specify the start point.

4 Specify the second point.

5 Specify the next point.

**TIP** Edit the curve by moving any of the three points.

### Quick Reference

Ribbon

Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

🌰 Fixed Curve (Three Point)

## Adding a Fixed Curve with Two Points and a Direction at First Point

Add a fixed three-point curve defined by specified start and end points and a direction at the start point.

The result is a three-point curve. The specified curve direction (3) determines the location of the pass-through point between the start (1) and end (2) points.

**To add a fixed curve (two points and a direction at first point)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏️ .

2   On the Alignment Layout Tools toolbar, click 🔼 Fixed Curve (Two Points And Direction At First Point).

3   Specify the start point.

4   Specify the second point.

5   Specify the direction at the start point, or specify either a bearing or azimuth.

> **TIP** Edit the curve by moving any of the three points.

## Quick Reference

Ribbon

  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏️

Menu

  Alignments menu ➤ Edit Alignment Geometry

Command Line

  EditAlignment

Alignment Layout Tools Toolbar icons

  🔼 Fixed Curve (Two Points And Direction At First Point)

## Adding a Fixed Curve with Two Points and a Direction at Second Point

Add a fixed curve defined by specified start and end points and a direction at the end point.

The result is a three-point curve. The specified curve direction (3) determines the location of the pass-through point between the start (1) and end (2) points.
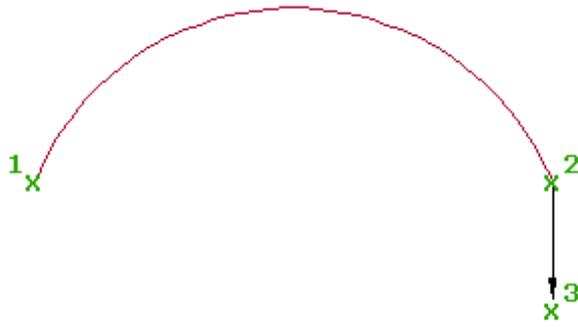
**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).
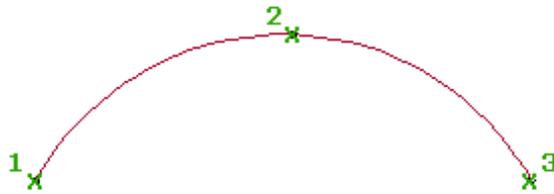
**To add a fixed curve (two points and a direction at second point)**

1　Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✐ .

2　On the Alignment Layout Tools toolbar, click ⌒ Fixed Curve (Two Points And Direction At Second Point).

3　Specify the start point.

4　Specify the next point. A preview of the curve is displayed.

5　Specify the direction at the second point or specify a bearing or azimuth.

> **TIP** Edit the curve by moving any of the three points.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✐

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons
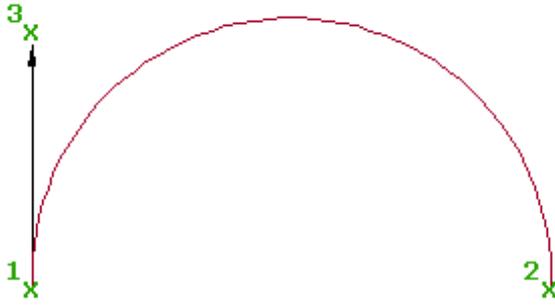
⌒ Fixed Curve (Two Points And Direction At Second Point)

## Adding a Fixed Curve with Two Points and Radius

Add a fixed curve defined by specified radius, direction, and start and end points.

The result is a three-point curve. The pass-through point between the end points (1, 4) is located at the curve mid point.
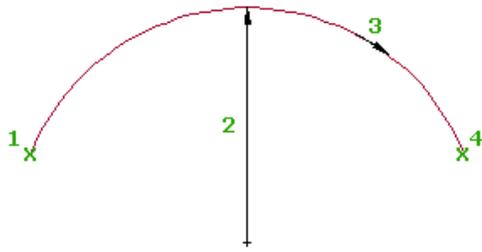
**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a fixed curve (two points and radius)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ⟋ .

2   On the Alignment Layout Tools toolbar, click ⌒ Fixed Curve (Two Points And Radius).

3   Specify the start point.

4   Specify a radius.

    Specify a new value, or press Enter to accept the value that is displayed on the command line.

    ■   If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

    ■   If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5   Specify the curve direction, either clockwise or counter-clockwise. A preview of the curve is displayed.

6   Specify the end point.

    **TIP** Edit the curve by moving any of the three points.

## Quick Reference

Ribbon

  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ⟋

Menu

  Alignments menu ➤ Edit Alignment Geometry

Command Line

  EditAlignment

Alignment Layout Tools Toolbar icons

  ⌒ Fixed Curve (Two Points And Radius)

## Adding a Fixed Curve from an Entity End to a Through Point

Add a fixed curve from the end of an existing entity to a specified end point

The result is a three-point curve. When the curve is created, it is tangent to the end of the selected entity, but if you edit either the curve or the attachment entity, tangency is not maintained.
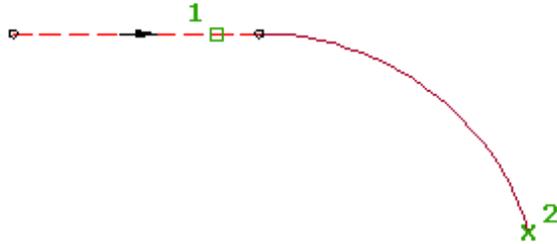


---

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

**To add a fixed curve (from entity end, through point)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ⌒ Fixed Curve (From Entity End, Through Point).

3 Select the entity for the start point and the direction.

4 Specify the next point.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⌒ Fixed Curve (From Entity End, Through Point)

## Adding a Fixed Curve from Center Point with Radius

Add a full, fixed circle defined by a specified center point, direction, and radius.

The result is a circle with four pass-through points. The entity position is not affected by edits to adjacent entities.

---

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a fixed curve (center point and radius)**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2  On the Alignment Layout Tools toolbar, click ⟲ Fixed Curve (Center Point And Radius).

3  Specify the center point.

4  Specify the curve direction: either clockwise or counter-clockwise.

5  Specify a radius.

   Specify a new value, or press Enter to accept the value that is displayed on the command line.

   ■  If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

   ■  If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

   A preview of the curve is displayed. A fixed circle with a center point and a radius is created.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

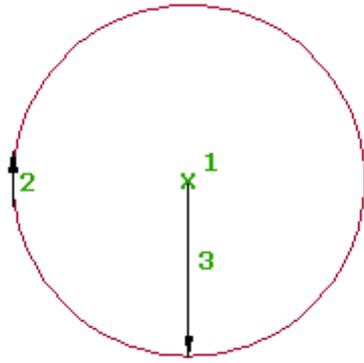Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⟲ Fixed Curve (Center Point And Radius)

## Adding a Fixed Curve with a Center Point and Through Point

Add a full, fixed circle defined by a specified center point, direction, and pass-through point.

The result is a circle with four pass-through points. The entity position is not affected by edits to adjacent entities.
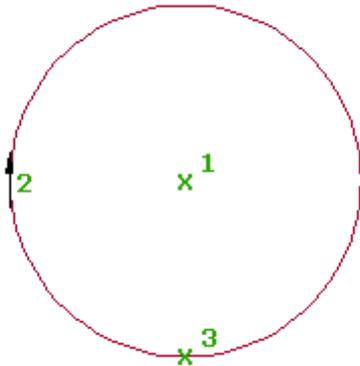


**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a fixed curve (center point, through point)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ⊙ Fixed Curve (Center Point, Through Point).

3 Specify the center point.

4 Specify the curve direction: either clockwise or counter-clockwise.

5 Specify a pass-through point. A preview of the curve is displayed.

**TIP** Edit the curve by moving the center point or the pass-through point.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⊙ Fixed Curve (Center Point, Through Point)

## Adding a Fixed Curve with a Through Point, Direction at Point, and Radius

Add a full, fixed circle defined by a specified pass-through point, direction at the pass-through point, curve direction, and radius.

The result is a circle with four pass-through points. The entity position is not affected by edits to adjacent entities.
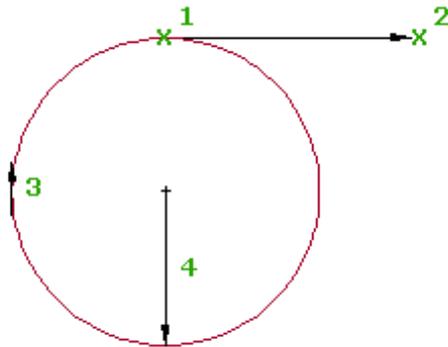


**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a fixed curve (through point, direction at point, and radius)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click Fixed Curve (Through Point, Direction At Point And Radius).

3 Specify the pass-through point.

4 Specify the direction at the pass-through point or specify a bearing or azimuth.

5 Specify the curve direction: either clockwise or counter-clockwise.

6 Specify a radius.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

■ If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

■ If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

A curve preview is displayed.

**TIP** Edit the curve by moving the center point or using the radius grips.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

 Fixed Curve (Through Point, Direction At Point And Radius)

## Adding a Fixed Curve by Best Fit

Add the most probable fixed curve through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

The result is a three-point curve. Regression points can be added, removed, or modified after the entity has been created.



**See also:**

■ Editing Best Fit Alignment Entities (page 1003)

**To create a fixed curve by best fit from AutoCAD Civil points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor  .

2 On the Alignment Layout Tools toolbar, click  Fixed Curve - Best Fit.

3 In the Curve By Best Fit Dialog Box (page 1989), select From COGO Points.

4 Select three or more AutoCAD Civil points. Enter G to select a point group or N to enter points by number.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**NOTE** You must select at least three non-collinear regression points.

5 Press Enter to complete the command.

6 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the curve:

■ Click 🖫 to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To create a fixed curve by best fit from AutoCAD points**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

**2** On the Alignment Layout Tools toolbar, click 📐 Fixed Curve - Best Fit.

**3** In the Curve By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

**4** Select three or more AutoCAD points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

NOTE You must select at least three non-collinear regression points.

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the curve:

■ Click 🖫 to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To create a fixed curve by best fit from existing entities**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

**2** On the Alignment Layout Tools toolbar, click 📐 Fixed Curve - Best Fit.

**3** In the Curve By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**4** Select one or more of the entities listed on the command line.

NOTE You can select several types of entities listed on the command line.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

NOTE You must select at least three non-collinear regression points.

**5** If you selected a profile object, specify the starting and ending station on the Specify Station Range Dialog Box (page 1992).

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the curve:

■ Click ![save icon] to create the curve and keep the Regression Data vista open.

■ Click ![check icon] to create the curve and close the Regression Data vista.

**To create a fixed curve by best fit by clicking on screen**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon] .

**2** On the Alignment Layout Tools toolbar, click ![icon] Fixed Curve - Best Fit.

**3** In the Curve By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**4** Select a starting point and at least two other points.

---
**NOTE** You can use OSNAP or transparent commands to select points.

---

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

---
**NOTE** You must select at least three non-collinear regression points.

---

**5** Press Enter to complete the command.

**6** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**7** Create the curve:

■ Click ![save icon] to create the curve and keep the Regression Data vista open.

■ Click ![check icon] to create the curve and close the Regression Data vista.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

 Fixed Curve - Best Fit

 Edit Best Fit Data For All Entities

Dialog Box

Curve By Best Fit Dialog Box (page 1989)
Regression Data Vista (page 1990)

## Adding Floating Curves

Add floating curves to build constraint-based alignment geometry for the areas of your design that require maintaining tangency.

## Adding a Floating Curve with a Radius and a Through Point to an Entity

Add a floating curve, defined by a specified radius and angle range, from an existing entity to a specified end point.

This curve supports both compound and reverse solutions.
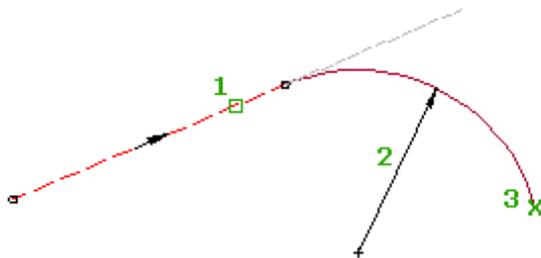


---

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

The following illustration shows a floating curve attached to a fixed line and two possible solutions for a given radius and point:
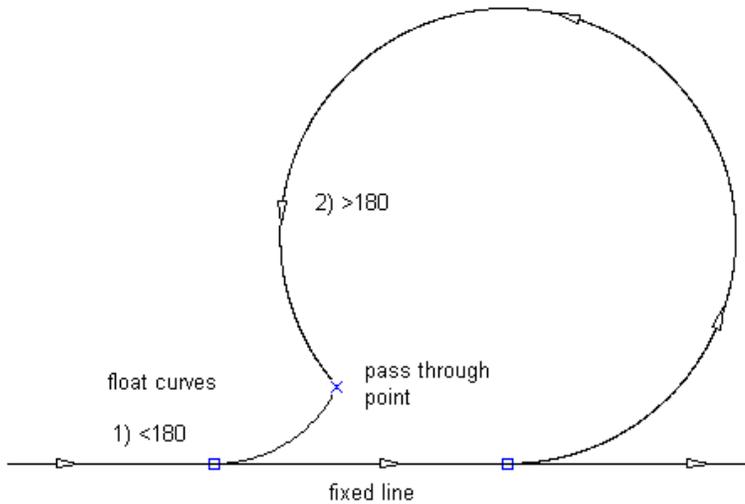
The following illustration shows a floating compound curve attached to a curve and four possible solutions for a given radius and pass-through point. The illustration also shows variables for greater than and less than 180 degrees and compound and reverse variables:
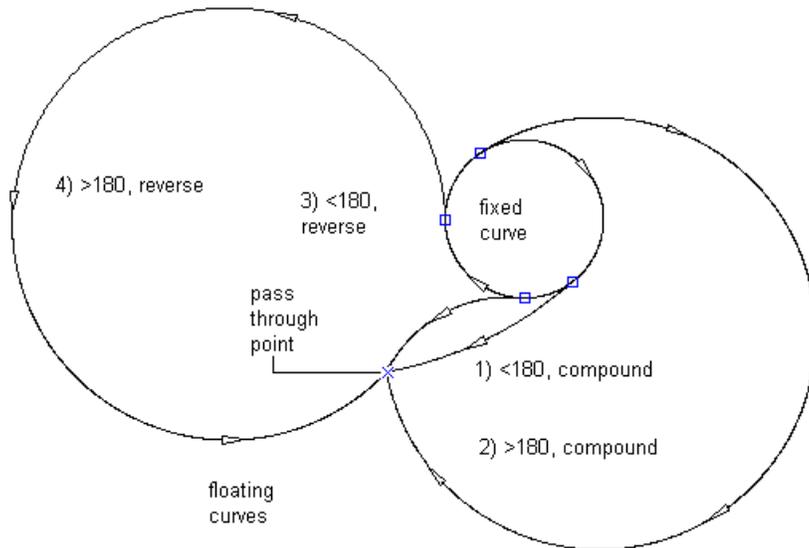


**To add a floating curve (from entity, radius, through point)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click  Floating Curve (From Entity, Radius, Through Point).

3 Select the end of the entity to which you want to add the floating curve.

4 Specify a radius.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

■ If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

**5** Specify whether the curve solution angle is either greater than or less than 180 degrees. A preview curve is displayed.

**NOTE** If the attachment entity is a curve, you are prompted to specify either a compound curve or a reverse curve.

**6** Specify the pass-through point.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

Floating Curve (From Entity, Radius, Through Point)

## Adding a Floating Curve with a Through Point to an Entity End

Add a floating curve from the end of an existing entity to a specified pass-through point.

The attachment entity (1) and pass-through point (2) determine the curve radius. You cannot edit the radius for this type of curve.
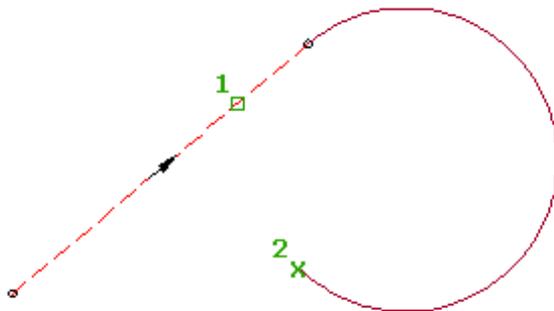


The result is a floating curve that is always tangent to the entity it is attached to.

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a floating curve (from entity end, through point)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor  .

2   On the Alignment Layout Tools toolbar, click  Floating Curve (From Entity End, Through Point).

3   Select the end of the entity to which you want to add the floating curve.

4   Specify the end point. A curve preview is displayed.

## Quick Reference

Ribbon

  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 

Menu

  Alignments menu ➤ Edit Alignment Geometry

Command Line

  EditAlignment

Alignment Layout Tools Toolbar icons

  Floating Curve (From Entity End, Through Point)

## Adding a Floating Curve with a Direction, Radius, and Length to an Entity End

Add a floating curve entity that is defined by the entity end to which it is attached, the radius, and the length by specifying the entity end where you want to add the curve.

This curve is not defined by a pass-through point, so the entire curve moves with the entity to which it is attached.
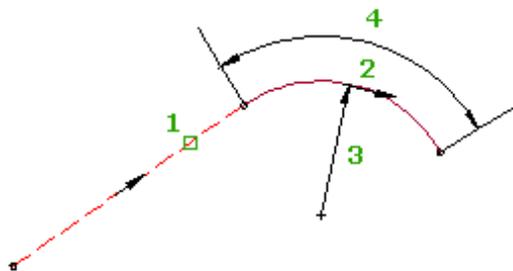


**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a floating curve attached (from entity end, radius, length)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor  .

**2** On the Alignment Layout Tools toolbar, click ⌖ Floating Curve (From Entity End, Radius, Length).

**3** Select the end of the entity to which you want to add the floating curve.

**4** Specify the curve direction: either clockwise or counter-clockwise.

**5** Specify a radius.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

- If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

**6** Specify the length or [deltaAngle/Tanlen/Chordlen/midOrd/External].

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⌖ Floating Curve (From Entity End, Radius, Length)

## Adding a Floating Curve with a Through Point and a Direction to an Entity

Add a floating curve with a specified pass-through point and direction at the pass-through point to an existing entity.

The attachment entity (1), pass-through point (2), and direction (3) determine the curve attachment point and radius. You cannot edit the radius for this type of curve.
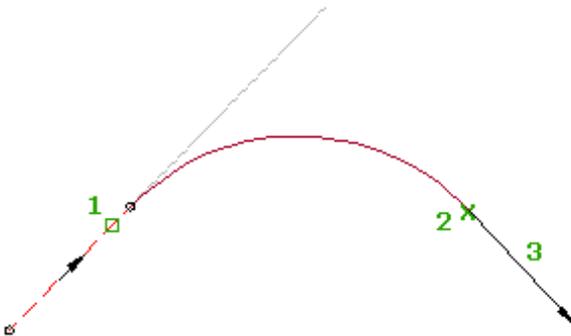


The result is a floating curve that is always tangent to the entity it is attached to.

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**To add a floating curve (from entity, through point, direction at point)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎.

2 On the Alignment Layout Tools toolbar, click ✎ Floating Curve (From Entity, Through Point, Direction At Point).

3 Specify the entity to which you want to add the floating curve.

4 Specify the end point.

5 Specify the direction at the end point or specify bearing or azimuth.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

✎ Floating Curve (From Entity, Through Point, Direction At Point)

## Adding a Floating Curve by Best Fit

Add the most probable floating curve through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

The location of the curve start point on the attachment entity (1) is determined by the path through the selected regression points (2).

**See also:**

■ Editing Best Fit Alignment Entities (page 1003)

**To create a floating curve by best fit using AutoCAD Civil points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ⬚ .

2 On the Alignment Layout Tools toolbar, click ⬚ Floating Curve - Best Fit.

3 Select an entity for the start point and direction.

If you select the beginning of the entity, the best fit curve will precede the entity. If you select the end of the entity, the best fit curve will succeed the entity.

4 In the Curve By Best Fit Dialog Box (page 1989), select From COGO Points.

5 Select two or more AutoCAD Civil points. Enter G to select a point group or N to enter points by number.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**NOTE** You must select at least two non-collinear regression points.

6 Press Enter to complete the command.

7 In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

8 Create the curve:

■ Click ⬚ to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To create a floating curve by best fit using AutoCAD points**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ⬚ .

2 On the Alignment Layout Tools toolbar, click ⬚ Floating Curve - Best Fit.

3 Select an entity for the start point and direction.

If you select the beginning of the entity, the best fit curve will precede the entity. If you select the end of the entity, the best fit curve will succeed the entity.

4 In the Curve By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

5 Select two or more AutoCAD points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**NOTE** You must select at least two non-collinear regression points.

6 Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the curve:

■ Click [icon] to create the curve and keep the Regression Data vista open.

■ Click [icon] to create the curve and close the Regression Data vista.

**To create a floating curve by best fit using existing entities**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor [icon] .

**2** On the Alignment Layout Tools toolbar, click [icon] Floating Curve - Best Fit.

**3** Select an entity for the start point and direction.

If you select the beginning of the entity, the best fit curve will precede the entity. If you select the end of the entity, the best fit curve will succeed the entity.

**4** In the Curve By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**5** Select one or more of the entities listed on the command line.

**NOTE** You can select several types of entities listed on the command line.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**NOTE** You must select at least two non-collinear regression points.

**6** If you selected a profile object, specify the starting and ending station on the Specify Station Range Dialog Box (page 1992).

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the curve:

■ Click [icon] to create the curve and keep the Regression Data vista open.

■ Click [icon] to create the curve and close the Regression Data vista.

**To create a floating curve by best fit by clicking on screen**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor [icon] .

**2** On the Alignment Layout Tools toolbar, click [icon] Floating Curve - Best Fit.

**3** Select an entity for the start point and direction.

If you select the beginning of the entity, the best fit curve will precede the entity. If you select the end of the entity, the best fit curve will succeed the entity.

**4** In the Curve By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**5** Select at least two points.

---

**NOTE** You can use OSNAP or transparent commands to select points.

---

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

---

**NOTE** You must select at least two non-collinear regression points.

---

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the curve:

- Click ![save icon] to create the curve and keep the Regression Data vista open.

- Click ![check icon] to create the curve and close the Regression Data vista.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

![icon] Floating Curve - Best Fit

![icon] Edit Best Fit Data For All Entities

Dialog Box

Curve By Best Fit Dialog Box (page 1989)
Regression Data Vista (page 1990)

## Adding Free Curves

Add free curves to build constraint-based alignment geometry for the areas of your design that require maintaining tangency.

These entity types are very similar to the AutoCAD "fillet" command, but give you added control.

## Adding a Free Curve Fillet with a Specified Radius Between Two Entities

Add a free curve, with a specified angle range and radius, between two entities.

This entity is defined by the attachment entities (1, 2), radius (3), solution angle, and whether the required solution is compound or reverse. The solution can vary greatly, depending on how you define the constraints.
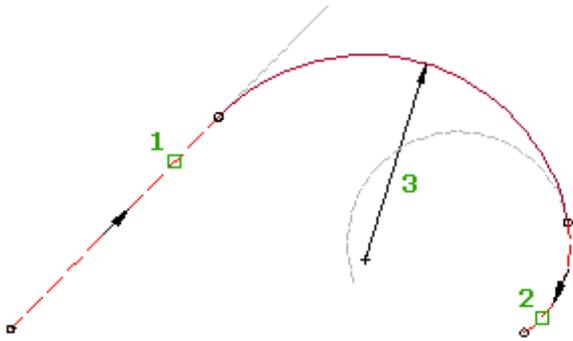
The following illustration shows a free curve fillet between two fixed lines. There are two solutions based on the included angle of the curve:



The following illustration shows a free curve fillet between two fixed clockwise curves:

2) compound & >180

1) compound & <180

cw
fixed
curve

cw
fixed
curve

3) reverse & <180

4) reverse & >180
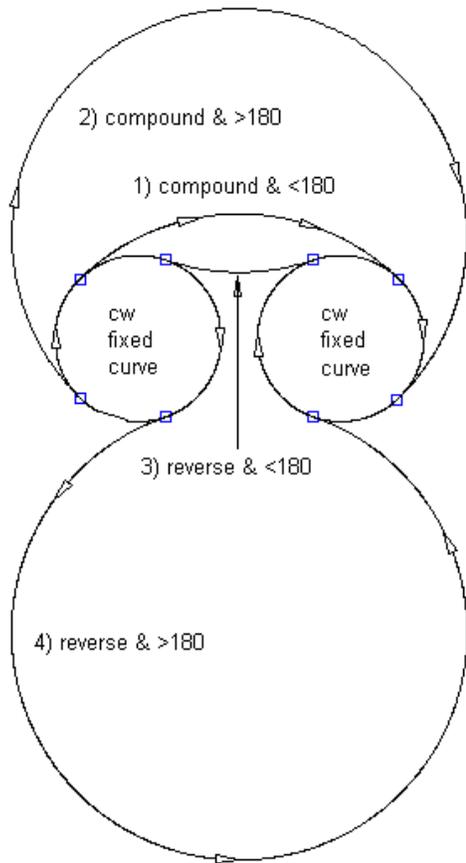
The following illustration shows a free curve fillet between a fixed clockwise curve and a fixed counter-clockwise curve:

2) compound to curve 1,
reverse to curve 2 & >180

1) compound to curve 1,
reverse to curve 2 & <180

cw
fixed
curve 1

ccw
fixed
curve 2

3) reverse to curve 1,
compound to curve 2
& <180

4) reverse to curve 1,
compound to curve 2
& >180

The following illustration shows a free curve fillet between an overlapping clockwise curve and a counter-clockwise curve:



2) compound to curve 1,
reverse to curve 2 & >180

3) reverse to curve 1,
compound to curve 2 & <180

ccw
fixed
curve 1

cw
fixed
curve 2

1) compound to curve 1,
reverse to curve 2 & <180

4) reverse to curve 1,
compound to curve 2 & >180
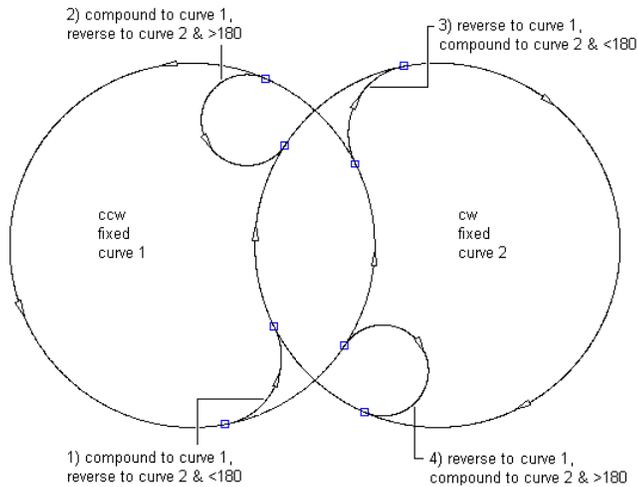
The following illustration shows a free curve fillet between a line and a clockwise curve (on the left) with a specified radius:
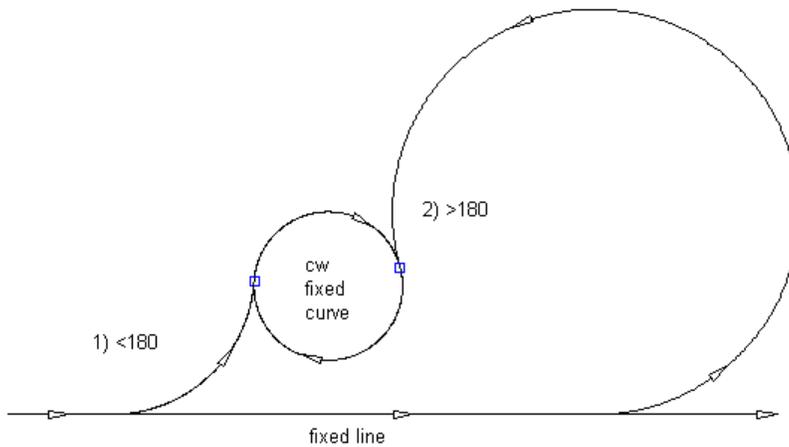
The following illustration shows a free curve fillet between a line and a counter-clockwise curve (on the left) with a specified radius:
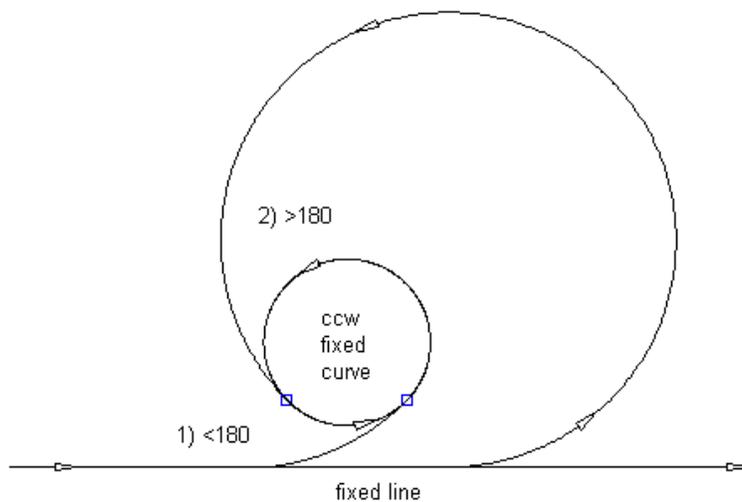
**To add a free curve fillet (between two entities, radius)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click  Free Curve Fillet (Between Two Entities, Radius).

3 Specify the entity (line, curve) from which you want to add the free curve.

4 Specify the entity (line, curve) to which you want to add the free curve.

   **NOTE** The entity before and the entity after must have the same direction.

5 Specify whether the curve solution angle is either greater than or less than 180 degrees.

6 Specify the radius or [curveLen/Tanlen/Chordlen/midOrd/External].

   Specify a new value, or press Enter to accept the value that is displayed on the command line.

   ■ If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

■ If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

To display a curve preview you must pick a point in the drawing or enter a coordinate value, then a rubber band is drawn to the cursor from the selected point and the curve preview is displayed.

7 Specify the second point. You can enter a 2D coordinate point or use a transparent command. For more information, see Transparent Commands (page 1581).

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

![icon] Free Curve Fillet (Between Two Entities, Radius)

## Adding a Free Curve Fillet with a Through Point Between Two Entities

Add a free curve, with a specified pass-through point, between two entities.

You can add this entity between two lines, two curves, or a line and a curve. The attachment entities (1, 2) and pass-through point (3) determine the curve radius and length. You cannot edit the radius and length for this type of curve.
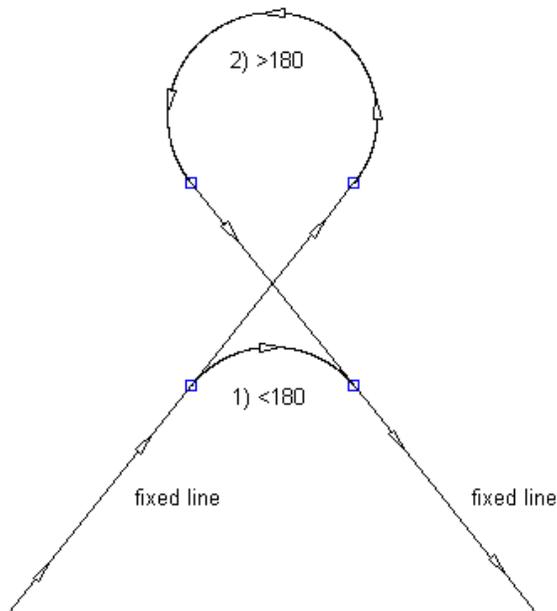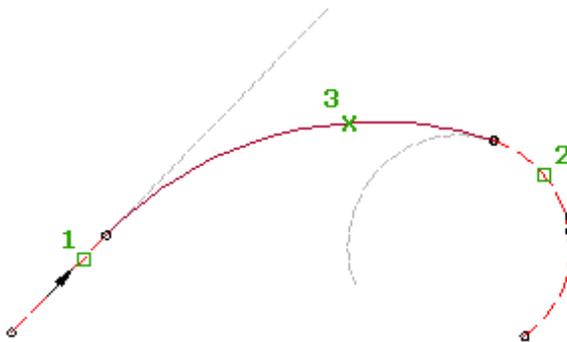


**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).
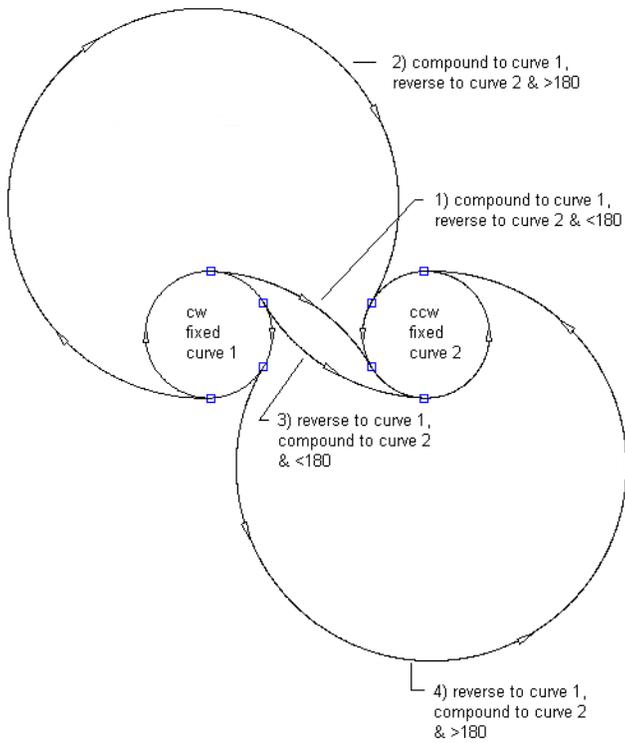
**To add a free curve fillet (between two entities, through point)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon] .

**2** On the Alignment Layout Tools toolbar, click ⌃ Free Curve Fillet (Between Two Entities, Through Point).

**3** Select the entity from which you want to add the free curve.

**4** Select the entity to which you want to attach the free curve. A curve preview is displayed.

**5** Specify the pass-through point. You can enter a 2D coordinate point or use a transparent command. For more information, see Transparent Commands (page 1581).

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⌃ Free Curve Fillet (Between Two Entities, Through Point)

## Adding a Free Curve by Best Fit

Add the most probable free curve between two entities through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

The location of the curve start and end points on the attachment entities (1,2) is determined by the path through the selected regression points (3).
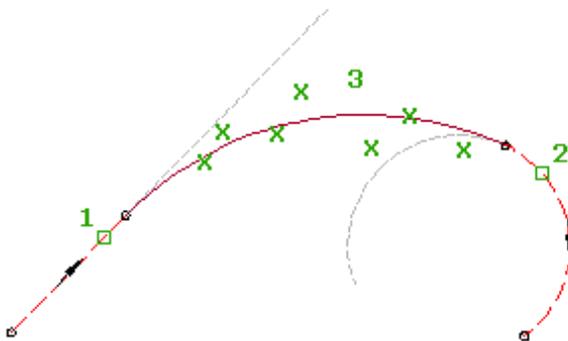


**See also:**

■ Editing Best Fit Alignment Entities (page 1003)

**To create a free curve by best fit using AutoCAD Civil points**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

**2** On the Alignment Layout Tools toolbar, click [icon] Free Curve - Best Fit.

**3** Select the entity from which you want to attach the free curve (the First Entity).

**4** Select the entity to which you want to add the free curve (the Next Entity).

---
**NOTE** The entity before and the entity after must have the same direction.

---

**5** In the Curve By Best Fit Dialog Box (page 1989), select From COGO Points.

**6** Select one or more AutoCAD Civil points. Enter G to select a point group or N to enter points by number.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the curve:

■ Click [icon] to create the curve and keep the Regression Data vista open.

■ Click [icon] to create the curve and close the Regression Data vista.

**To create a free curve by best fit using AutoCAD points**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor [icon] .

**2** On the Alignment Layout Tools toolbar, click [icon] Free Curve - Best Fit.

**3** Select the entity from which you want to attach the free curve (the First Entity).

**4** Select the entity to which you want to add the free curve (the Next Entity).

---
**NOTE** The entity before and the entity after must have the same direction.

---

**5** In the Curve By Best Fit Dialog Box (page 1989), select From AutoCAD Points.

**6** Select one or more AutoCAD points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the curve:

■ Click [icon] to create the curve and keep the Regression Data vista open.

■ Click [icon] to create the curve and close the Regression Data vista.

**To create a free curve by best fit using existing entities**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

**2** On the Alignment Layout Tools toolbar, click  Free Curve - Best Fit.

**3** Select the entity from which you want to attach the free curve (the First Entity).

**4** Select the entity to which you want to add the free curve (the Next Entity).

> **NOTE** The entity before and the entity after must have the same direction.

**5** In the Curve By Best Fit Dialog Box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**6** Select one or more of the entities listed on the command line.

> **NOTE** You can select several types of entities listed on the command line.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**7** If you selected a profile object, specify the starting and ending station in the Specify Station Range Dialog Box (page 1992).

**8** Press Enter to complete the command.

**9** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**10** Create the curve:

- Click  to create the curve and keep the Regression Data vista open.

- Click  to create the curve and close the Regression Data vista.

**To create a free curve by best fit by clicking on screen**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

**2** On the Alignment Layout Tools toolbar, click  Free Curve - Best Fit.

**3** Select the entity from which you want to attach the free curve (the First Entity).

**4** Select the entity to which you want to add the free curve (the Next Entity).

> **NOTE** The entity before and the entity after must have the same direction.

**5** In the Curve By Best Fit Dialog Box (page 1989), select By Clicking On The Screen.

**6** Select one or more points on screen.

> **NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data (page 1990) vista to make changes to the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the curve:

■ Click ![save icon] to create the curve and keep the Regression Data vista open.

■ Click ![checkmark icon] to create the curve and close the Regression Data vista.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ![icon]

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

![icon] Free Curve - Best Fit

![icon] Edit Best Fit Data For All Entities

Dialog Box

Curve By Best Fit Dialog Box (page 1989)
Regression Data Vista (page 1990)

# Adding Spirals

Use the Spiral commands to add fixed, free, or floating spirals to the alignment geometry.

## Spiral Definitions

Various transition curves are used in civil engineering to gradually introduce curvature and superelevation between both tangents and circular curves as well as between two circular curves with different curvature.

In its relationship to other tangents and curves, each spiral is either an incurve (page 2476) or an outcurve (page 2479).

The two most commonly used parameters by engineers in designing and setting out a spiral are L (spiral length) and R (radius of circular curve).

The following illustration shows the various parameters of a spiral:
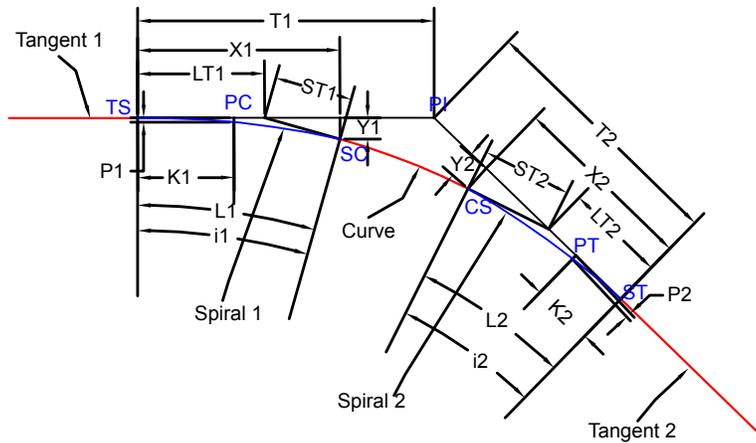
| Spiral Parameter | Description |
|---|---|
| i1 | The central angle of spiral curve L1, which is the spiral angle. |
| i2 | The central angle of spiral curve L2, which is the spiral angle. |
| T1 | The total tangent distance from PI to TS. |
| T2 | The total tangent distance from PI to ST. |
| X1 | The tangent distance at SC from TS. |
| X2 | The tangent distance at CS from ST. |
| Y1 | The tangent offset distance at SC from TS. |
| Y2 | The tangent offset distance at CS from ST. |
| P1 | The offset of the initial tangent into the PC of the shifted curve. |
| P2 | The offset of the initial tangent out to the PT of the shifted curve. |
| K1 | The abscissa of the shifted PC referred to the TS. |
| K2 | The abscissa of the shifted PT referred to the ST. |
| LT1 | The long tangent spiral in. |
| LT2 | The long tangent spiral out. |
| ST1 | The short tangent spiral in. |
| ST2 | The short tangent spiral out. |
| | Other Spiral Parameters |

| A1 | The A value equals the square root of the spiral length multiplied by the radius. A measure of the flatness of the spiral. |
|---|---|
| A2 | The A value equals the square root of the spiral length multiplied by the radius. A measure of the flatness of the spiral. |

Formula

$$\text{Flatness of spiral} = A = \sqrt{LR}$$

### Compound Spiral

Compound spirals provide a transition between two circular curves with different radii. As with the simple spiral, this allows for continuity of the curvature function and provides a way to introduce a smooth transition in superelevation.

### Clothoid Spiral

While AutoCAD Civil supports several spiral types, the clothoid spiral is the most commonly used spiral type. The clothoid spiral is used world wide in both highway and railway track design.

First investigated by the Swiss mathematician Leonard Euler, the curvature function of the clothoid is a linear function chosen such that the curvature is zero (0) as a function of length where the spiral meets the tangent. The curvature then increases linearly until it is equal to the adjacent curve at the point where the spiral and curve meet.

Such an alignment provides for continuity of the position function and its first derivative (local azimuth), just as a tangent and curve do at a Point of Curvature (PC). However, unlike the simple curve, it also maintains continuity of the second derivative (local curvature), which becomes increasingly important at higher speeds.

Formula

Clothoid spirals can be expressed as: $\theta = \dfrac{l^2}{2RL}$

Flatness of spiral: $A = \sqrt{LR}$

Total angle subtended by spiral: $i_s = \dfrac{L}{2R}$

Tangent distance at spiral-curve point from tangent-spiral point is:

$$X = L*[1 - \frac{L^2}{40R^2} + \frac{L^4}{3456R^4} - ...]$$

Tangent offset distance at spiral-curve point from tangent-spiral point is:

$$Y = \frac{L^2}{6R}[1 - \frac{L^2}{56R^2} + \frac{L^4}{7040R^4} - ...]$$

### Bloss Spiral

Instead of using the clothoid, the Bloss spiral with the parabola of fifth degrees can be used as a transition. This spiral has an advantage over the clothoid in that the shift P is smaller and therefore there is a longer transition, with a larger spiral extension (K). This factor is important in rail design.

Formula

Bloss spirals can be expressed as:

$$\theta = \frac{l^3}{RL^2} - \frac{l^4}{2RL^3}$$

Other key expressions:

Tangent distance at spiral-curve point from tangent-spiral point is:

$$X = L - \frac{L^3}{43.8261R^2} + \frac{L^5}{3696.63R^4}$$

Tangent offset distance at spiral-curve point from tangent-spiral point is:

$$Y = \frac{3L^2}{20R} - \frac{L^4}{363.175R^3}$$

## Sinusoidal Curves

These curves represent a consistent course of curvature and are applicable to transition from 0 through 90 degrees of tangent deflections. However, sinusoidal curves are not widely used because they are steeper than a true spiral and are therefore difficult to tabulate and stake out.

Formula

Sinusoidal curves can be expressed as:

$$\theta = \frac{l^2}{2RL} + \left(\frac{L}{4\pi^2 R}\right)\left[\cos\left(\frac{2\pi l}{L}\right) - 1\right]$$

Differentiating with *l* we get an equation for *l*/*r*, where *r* is the radius of curvature at any given point:

$$r = \frac{2\pi LR}{2\pi l - L * SIN\left(\frac{2\pi l}{L}\right)}$$

## Sine Half-Wavelength Diminishing Tangent Curve

This form of equation is commonly used in Japan for railway design. This curve is useful in situations where you need an efficient transition in the change of curvature for low deflection angles (in regard to vehicle dynamics.)

Formula

Sine Half-Wavelength Diminishing Tangent curves can be expressed as:

$$y = \frac{X^2}{R}\left[\frac{a^2}{4} - \frac{1}{2\pi^2}\{1 - \cos a\pi\}\right]$$

$$a = \frac{x}{X}$$

where and x is the distance from the start to any point on the curve and is measured along the (extended) initial tangent; *X* is the total X at the end of the transition curve.

Other key expressions:

Tangent distance at spiral-curve point from tangent-spiral point is:

$$X = L - \left( \frac{2\pi^2 - 9}{48\pi^2} \right) * \frac{L^3}{R^2} = L - 0.0226689447 \frac{L^3}{R^2}$$

Tangent offset distance at spiral-curve point from tangent-spiral point is:

$$Y = \left[ \frac{1}{4} - \frac{1}{\pi^2} \right] * \frac{X^2}{R} = 0.14867881635766 \frac{X^2}{R}$$

**Cubic Spiral (JP)**

This spiral is developed for requirements in Japan. Some approximations of the clothoid have been developed to use in situations to accommodate a small deflection angle or a large radius. One of these approximations, used for design in Japan, is the Cubic Spiral (JP).

Formula

Cubic Spirals (JP) can be expressed as:

$$y = \frac{x^3}{6RX}$$

Where X = Tangent distance at spiral-curve point from tangent-spiral point

This formula can also be expressed as:

$$\tan \theta_s = \frac{x^2}{2RX}$$

Where $\theta$ is central angle the spiral (illustrated as i1 and i2in the illustration)

Other key expressions:

Tangent distance at spiral-curve point from tangent-spiral point is:

$$X = L * \left[ \frac{10}{10 + (\tan \theta_s)^2} \right]$$

Tangent offset distance at spiral-curve point from tangent-spiral point is:

$$TotalY = \frac{X^2}{6R}$$

**Cubic Parabolas**

Cubic parabolas converge less rapidly than cubic spirals, which makes their use popular in railway and highway design. While they are less accurate than cubic spirals, cubic parabolas are preferred by highway and railway engineers because they are expressed in Cartesian coordinates and are easy to set out in the field.

Formula

When $\theta$ -> zero -> we can assume that cos $\theta$ = 1, then $x = l$.

Further, if we assume that sin $\theta = \theta$ , then

$x = l$ and *TotalX = (approximately) L*

Substituting this approximation helps us obtain the following equation:

$$y = \frac{x^3}{6RL}$$

All other parameters are the same as the clothoid spiral.

Minimum Radius of Cubic Parabola

The radius at any point on a cubic parabola is:

$$r = \frac{\sqrt{RL}}{\sqrt{2\sin\theta\cos^5\theta}}$$

A cubic parabola attains minimum *r* at:

$$\tan\theta = \frac{1}{\sqrt{5}}$$

So $r_{min} = 1.39\sqrt{RL}$

A cubic parabola radius decreases from *infinity* to $r_{min} = 1.39\sqrt{RL}$ at 24 degrees, 5 minutes, 41 seconds and from then onwards starts to increase again. This makes cubic parabolas useless for deflections greater than 24 degrees.

**Bi-Quadratic (Schramm) Spirals**

Bi-quadratic (Schramm) spirals have low values of vertical acceleration. They contain two second-degree parabolas whose radii vary as a function of curve length.

Simple Curve Formula

Curvature of the first parabola:

$$\frac{1}{r} = \frac{2 * l^2}{RL^2} \quad \text{for} \quad 0 \le l \le \frac{L}{2}$$

Curvature of the second parabola:

$$\frac{1}{r} = \frac{4 * L * l - L^2 - 2 * l^2}{RL^2} \quad \text{for} \quad \frac{L}{2} \le l \le L$$

This curve is specified by the user-defined length (L) of the transition curve.

Compound Curve Formulas

Curvature of the first parabola:

$$\frac{1}{r} = \frac{1}{r_1} + \frac{2 * l^2}{L^2}\left(\frac{1}{R_2} - \frac{1}{R_1}\right) \text{ for } 0 \leq l \leq \frac{L}{2}$$

Curvature of the second parabola:

$$\frac{1}{r} = \frac{1}{r_2} - \left(\frac{L-l}{L}\right)^2 * \left(\frac{1}{R_2} - \frac{1}{R_1}\right) \text{ for } \frac{L}{2} \leq l \leq L$$

## Adding Fixed Spirals

Use the Alignment Layout Tools to add a fixed spiral to the end of an existing entity.

## Adding a Fixed Spiral

Add a fixed spiral to the end of a fixed or floating entity.

Depending on the desired solution, a fixed spiral can be defined by various parameters, which cannot be edited after creation. When the attachment entity (1) is edited, the defining parameter (2) does not change.
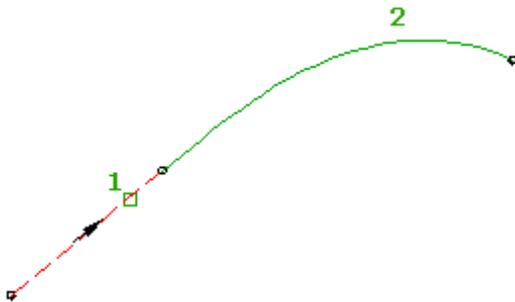


**To add a fixed spiral**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✐ .

2   On the Alignment Layout Tools toolbar, click ⊚ Fixed Spiral.

   The current spiral definition is displayed on the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861)

3   Select the entity for a start point and a direction.

4   Specify the spiral type: either Compound, Incurve, Outcurve, or Point.

5   Specify the curve direction: either Clockwise or Counterclockwise.

6   Specify a start radius by either picking two points in the drawing or entering a value on the command line.

   **NOTE** If the alignment has design criteria (page 866) applied to it, the minimum radius and spiral length values for the current design speed are displayed on the command line. Specify new values, or press Enter to accept the minimums.

**7**  Specify a length by either picking two points in the drawing or entering a value on the command line, or specify the A value.

**8**  Specify the end radius by picking two points in the drawing or entering a value on the command line.

## Quick Reference

Ribbon

  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

  Alignments menu ➤ Edit Alignment Geometry

Command Line

  EditAlignment

Alignment Layout Tools Toolbar icons

  ◎ Fixed Spiral

## Adding Free Spirals

Use the free spiral commands to create spiral solutions as a transition between two curves with different radii.

## Adding a Free Spiral Between Two Entities

Add a free spiral between two fixed or floating entities.

A free spiral has only one geometric solution, so you cannot specify its length or A value. The solution depends on the type of attachment entities (1, 2), and the order in which they are selected.



## Adding a Free Spiral Between Two Curves

Use this command in situations where you need to create a single compound spiral solution as a transition between two curves with different radii.

Because free spirals have only one geometric solution, you cannot specify length or A value. The program makes the calculation.

**IMPORTANT** Adding a free spiral between two curves is only supported for Clothoid and Bloss spiral types.

The following illustrations show a free spiral between two curves. The solution depends on the selection order of the curve that the spiral is being attached to. The first illustration shows a left solution, where the inner circle was selected first. The second illustration shows a right solution, where the inner circle was selected second:
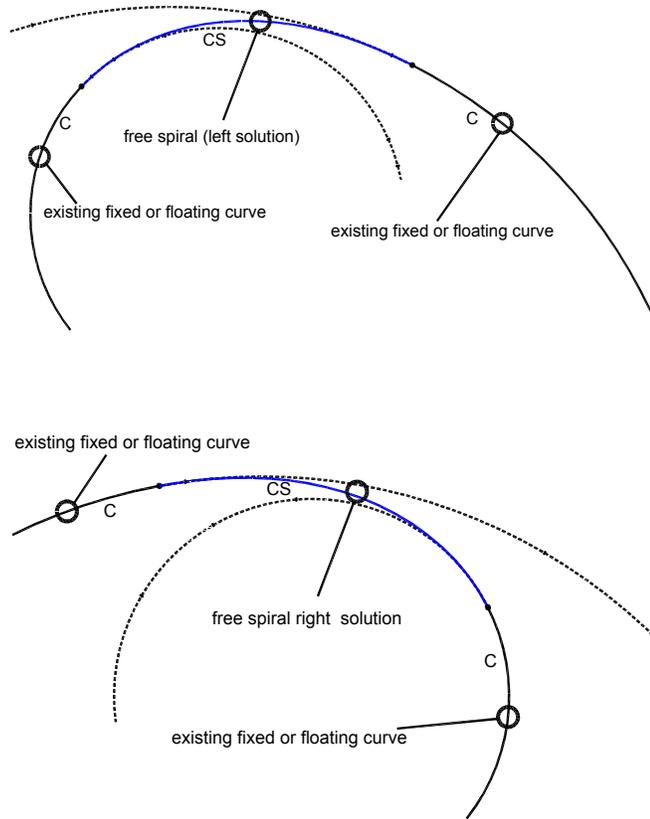




**To add a free spiral (between two curves)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ┆ Free Spiral (Between Two Entities).

   The current spiral definition is displayed on the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3 Select the entity before and the entity after to add the spiral to.

   The solution depends on the order in which you select the curves that the spiral is being attached to. For a solution where the spiral is coming from the left, select the inner circle first. For a solution where the spiral is coming from right, select the inner circle second.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

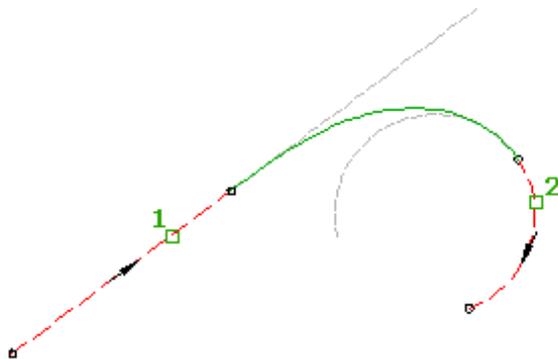Free Spiral (Between Two Entities)

## Adding a Free Spiral Between Two Lines

Add a free spiral between two lines in situations where you want the spirals to maintain tangency to the lines they are attached to.

For this solution, the spirals are simple, incurve, or outcurve.

The following illustration shows a free spiral between two lines. The possible solutions include:

- simple spiral (page 2482) incurve
- simple spiral outcurve



**To add a free spiral (between two lines)**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2  On the Alignment Layout Tools toolbar, click  Free Spiral (Between Two Entities).

   The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3  Select the entity before and the entity after to add the spiral to.

4  Specify the spiral type as either Incurve or Outcurve.

5  Specify the spiral length or the A value.

**Quick Reference**

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

Free Spiral (Between Two Entities)

## Adding a Free Spiral Between a Line and a Curve

Add a free spiral between a line and curve in situations where you want the spiral to maintain the radius of the curve it is attached to, even if the curve is edited.

For this solution, the spirals are simple because the radius of the spiral at the line end is infinite (as opposed to a compound spiral, where the radius at each end is finite). This solution between a line and a curve is valid for all spiral types because all spirals types support the simple spiral.

The two possible solutions depend on which entity you select first, either the line or the curve.

The following illustration shows a free spiral between a line and a curve, where the line was selected first:



**To add a free spiral (between a line and a curve)**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2  On the Alignment Layout Tools toolbar, click  Free Spiral (Between Two Entities).

   The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3  Select the line before and the curve after to add the spiral to.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

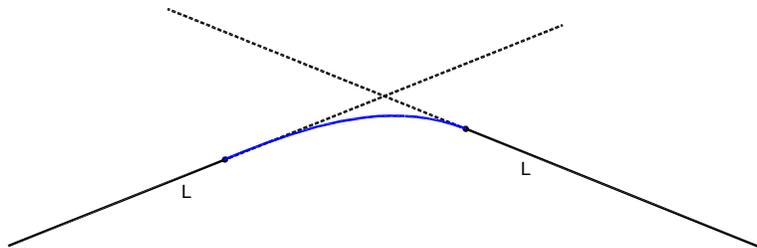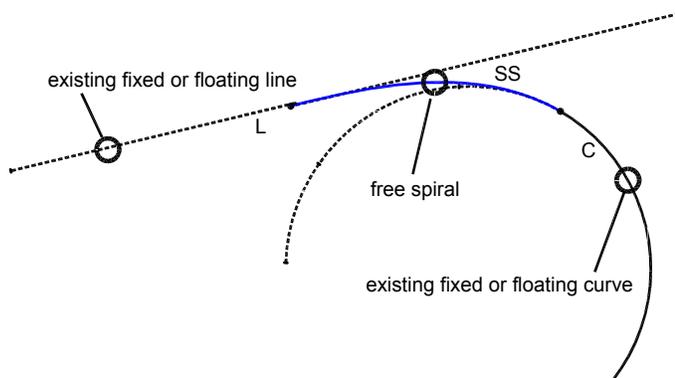Alignment Layout Tools Toolbar icons

⌐ Free Spiral (Between Two Entities)

## Adding a Free Compound Spiral-Spiral Between Two Curves

Add a free spiral-spiral group between two curves that are in the same direction but have different radii.

The spirals have finite radii on either end. Spiral parameters and attachment points are automatically calculated. If either of the attachment curves (1, 2) are edited, the spiral-spiral group maintains tangency.

By default, the two spirals have equal lengths and A values. Both values can be modified at the time of creation by specifying a ratio of A1/A2 or L1/L2.



The spiral radii are identical at the spiral-spiral intersection point. The radius value is determined by the total deflection angle and the two spiral lengths. The radius value cannot be edited.

**To add a free compound spiral-spiral (between two curves)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⌐ Free Compound Spiral-Spiral (Between Two Curves). The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3   Specify the entity before and the entity after to which you want to add the spiral.

4   Optionally, enter a value for the ratio of A1 to A2, or select Length and enter a value for the ratio of L1 to L2. Otherwise, the A values and lengths of each spiral will be equal.

**NOTE** If the curves are in the same direction, a compound solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution if the curves cross, or for concentric curves.

## Quick Reference

Ribbon

  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

  Alignments menu ➤ Edit Alignment Geometry

Command Line

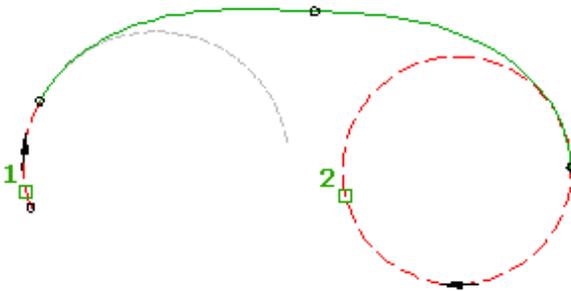  EditAlignment

Alignment Layout Tools Toolbar icons

  ⌇ Free Compound Spiral-Spiral (Between Two Curves)

## Adding a Free Reverse Spiral-Spiral Between Two Curves

Add a free reverse spiral-spiral group between two curves that are in different directions.

The two spirals bend in opposite directions. Spiral parameters and attachment points are automatically calculated. If either of the attachment curves (1, 2) are edited, the spiral-spiral group maintains tangency.
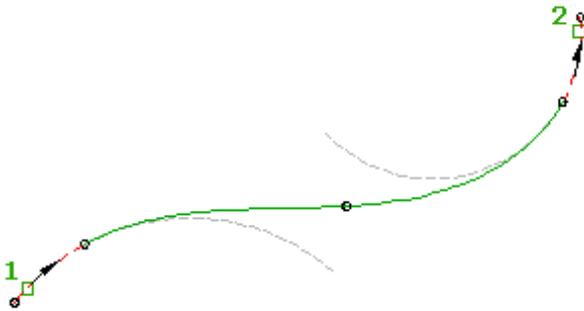


By default, the two spirals have equal lengths and A values. Both values can be modified at the time of creation by specifying a ratio of A1/A2 or L1/L2.

**To add a free reverse spiral-spiral (between two curves)**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2  On the Alignment Layout Tools toolbar, click ⌇ Free Reverse Spiral-Spiral (Between Two Curves).

   The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3  Specify the entity before and the entity after to which you want to add the spiral.

**4** Optionally, enter a value for the ratio of A1 to A2, or select Length and enter a value for the ratio of L1 to L2. Otherwise, the A values and lengths of each spiral will be equal.

> **NOTE** If the curves are in the same direction, a compound solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution if the curves cross, or for concentric curves.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖊

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⟡ Free Reverse Spiral-Spiral (Between Two Curves)

## Adding a Free Compound Spiral-Spiral Between Two Tangents

Add a free compound spiral-spiral group, with a specified parameter, between two tangents.

You specify either the length (3, 4) or A value for each spiral. The spiral radii match at the spiral-spiral intersection point. If either of the attachment tangents (1, 2) are edited, the spiral parameters that you specified do not change.
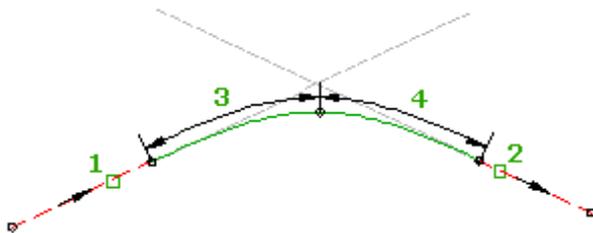


**To add a free compound spiral-spiral (between two tangents)**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖊 .

**2** On the Alignment Layout Tools toolbar, click ⟋ Free Compound Spiral-Spiral (Between Two Tangents).

**3** Select the tangent (the First Entity) from which you want to add the spiral group.

**4** Select the tangent (the Next Entity) to which you want to add the spiral group.

**5** Specify the length or A value of the first spiral (the spiral in).

Specify a new value, or press Enter to accept the value that is displayed on the command line.

- If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

6 Specify the length or A value of the second spiral (the spiral out).

7 Specify whether the solution is greater or less than 180 degrees.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⬔ Free Compound Spiral-Spiral (Between Two Tangents)

## Adding a Free Compound Spiral-Line-Spiral with Specified Spiral Lengths Between Two Curves

Add a free compound spiral-line-spiral group, with specified spiral lengths, between two curves.

The line length is automatically calculated. If either of the attachment curves (1, 2) are edited, the spiral lengths (3, 4) do not change.



**To add a free compound spiral-line-spiral (between two curves, spiral lengths)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ⬔ Free Compound Spiral-Line-Spiral (Between Two Curves, Spiral Lengths).

The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

**3** Specify the entity before and the entity after to which you want to add the spiral.

**4** Specify either the length or the A value for the spiral in and the spiral out.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

- If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

---

**NOTE** If the curves are in a different direction, a reverse solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution for concentric curves.

---

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✐

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⌓ Free Compound Spiral-Line-Spiral (Between Two Curves, Spiral Lengths)

## Adding a Free Reverse Spiral-Line-Spiral with Specified Spiral Lengths Between Two Curves

Add a free reverse spiral-line-spiral group, with specified spiral lengths, between two curves.

The two spirals bend in opposite directions. You specify the spiral lengths (2, 3). The line length is automatically calculated. If either of the attachment curves (1, 2) are edited, the spiral lengths do not change.
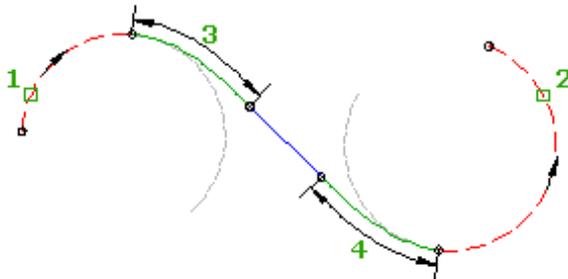
In this command you can specify the spiral parameters, but not the line length. Line length is calculated automatically.

**To add a free reverse spiral-line-spiral (between two curves, spiral lengths)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉 .

2   On the Alignment Layout Tools toolbar, click 👁 Free Reverse Spiral-Line-Spiral (Between Two Curves, Spiral Lengths).

The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3   Specify the entity before and the entity after to which you want to add the spiral.

4   Specify either the length or the A value for the spiral in and the spiral out.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

- If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

**NOTE** If the curves are in the same direction, a compound solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution for concentric curves.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

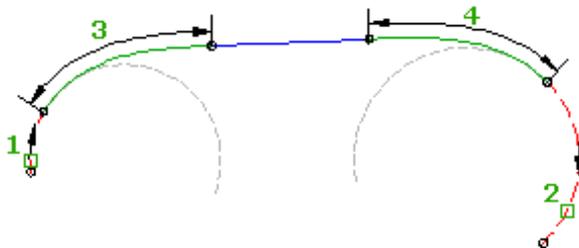👁 Free Reverse Spiral-Line-Spiral (Between Two Curves, Spiral Lengths)

## Adding a Free Compound Spiral-Line-Spiral with a Specified Line Length Between Two Curves

Add a free compound spiral-line-spiral group, with a specified line length, between two curves.

The spiral parameters are automatically calculated. If either of the attachment curves (1, 2) are edited, the line length (3) does not change.

**To add a free compound spiral-line-spiral (between two curves, line length)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ⌀ Free Compound Spiral-Line-Spiral (Between Two Curves, Line Length).

The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3 Specify the entity before and the entity after to which you want to add the spiral.

4 Specify the line length.

---

**NOTE** If the curves are in the same direction, a compound solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution for concentric curves.

---

## Quick Reference

Ribbon

   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

   Alignments menu ➤ Edit Alignment Geometry

Command Line

   EditAlignment

Alignment Layout Tools Toolbar icons

   ⌀ Free Compound Spiral-Line-Spiral (Between Two Curves, Line Length)

# Adding a Free Reverse Spiral-Line-Spiral with a Specified Line Length Between Two Curves

Add a free reverse spiral-line-spiral group, with a specified line length, between two curves.

The two spirals bend in opposite directions. You specify the line length (3). The spiral lengths are automatically calculated. If either of the attachment curves (1, 2) are edited, the line length does not change.

**To add a free reverse spiral-line-spiral (between two curves, line length)**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2 On the Alignment Layout Tools toolbar, click ⚬ Free Reverse Spiral-Line-Spiral (Between Two Curves, Line Length).

   The current spiral definition is displayed at the command line. For more information about changing the default spiral definition, see Specifying Curve and Spiral Settings (page 861).

3 Specify the entity before and the entity after to which you want to add the spiral.

4 Specify the line length.

---

**NOTE** If the curves are in the same direction, a compound solution is given. Enter Yes at the command line to accept the solution, or enter No to restart the command. There is no solution for crossing or concentric curves.

---

## Quick Reference

Ribbon

   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

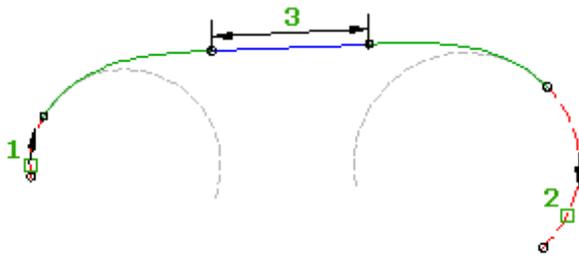   Alignments menu ➤ Edit Alignment Geometry

Command Line

   EditAlignment

Alignment Layout Tools Toolbar icons

   ⚬ Free Reverse Spiral-Line-Spiral (Between Two Curves, Line Length)

# Adding Lines with Spirals

Add entities that contain lines and transition spirals.

Lines and transition spirals can only be attached to fixed or floating entities.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

## Adding Floating Lines with Spirals

Add floating lines with spirals in situations where you need a spiral transition to a line that is defined by either a pass-through point or length.

## Adding a Floating Line with Spiral with a Through Point to a Curve

Add a floating spiral-line group, with a specified spiral length and pass-through point, to a curve.

When the attachment entity (1) is edited, the spiral length (2) and pass-through point (3) do not change. The line length and attachment point change to accommodate edits to the attachment curve.
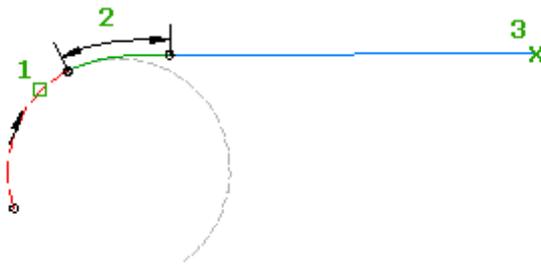


**To add a floating line with a spiral (from curve, through point)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⌒ Floating Line With Spiral (From Curve, Through Point).

    The current spiral definition is displayed at the command line.

3   Specify the curve end to attach to.

4   Specify either the spiral length or the A value, either by entering a numeric value, or selecting two points in the drawing.

    Specify a new value, or press Enter to accept the value that is displayed on the command line.

    ■   If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

    ■   If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5   Specify the pass through point.

**Quick Reference**

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

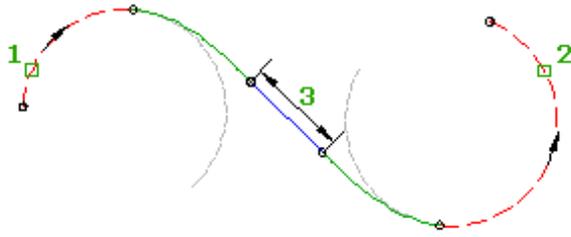Alignment Layout Tools Toolbar icons

⟋ Floating Line With Spiral (From Curve, Through Point)

## Adding a Floating Line with a Spiral with Specified Lengths to a Curve End

Add a floating spiral-line group, with specified spiral and line lengths, to a curve.

When the attachment entity (1) is edited, the spiral (2) and line (3) lengths do not change. You can adjust or move the attachment curve and the entire spiral-line group moves with it.



**To add a floating line with spiral (from curve end, length)**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⟋ Floating Line With Spiral (From Curve End, Length). The current spiral definition is displayed at the command line.

3   Specify the curve end to attach to.

4   Specify the spiral length or the A value, by either entering a numeric value or selecting two points.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

   ■   If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

   ■   If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5   Specify the line length.

**Quick Reference**

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

Floating Line With Spiral (From Curve End, Length)

# Adding Curves with Spirals

Add entities that contain curves and transition spirals.

This entity type can only be added to existing fixed or floating lines.

After clicking a button on the Alignment Layout Tools toolbar, follow the command prompts on the command line.

---
**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

## Adding Floating Curves with Spirals

Add floating curves with spirals in situations where you need a transition spiral and the curve geometry must pass through a specified point.

---
**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

## Adding a Floating Curve with a Spiral with a Specified Radius, Spiral Length, and Through Point to an Entity

Add a floating spiral-curve group, with a specified radius, spiral length, and pass-through point, to an existing line or curve entity.

When the attachment entity (1) is edited, the spiral length (3), solution angle (4), and curve radius (2) and pass-through point (5) do not change. The attachment point and curve length adjust to accommodate edits to the attachment curve.
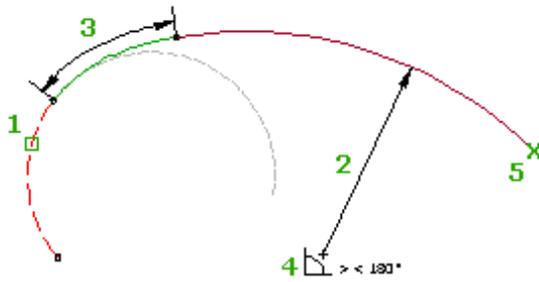
**To add a floating curve with spiral (from entity, through point) off a curve**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2   On the Alignment Layout Tools toolbar, click ⚓ Floating Curve With Spiral (From Entity, Radius, Through Point) to add a floating curve with a spiral off a curve.

    The current spiral definition is displayed on the command line.

3   Specify the curve entity to add to.

4   Specify the radius.

    Specify a new value, or press Enter to accept the value that is displayed on the command line.

    ■   If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

    ■   If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5   Specify either the spiral length or the A value.

6   Specify whether the curve solution angle is either greater than or less than 180 degrees.

7   Specify whether the curve is compound or reverse to the curve it is being added to.

8   Specify the pass through point.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

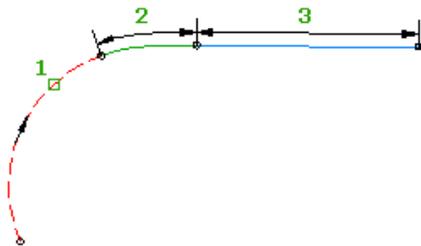Alignment Layout Tools Toolbar icons

⚓ Floating Curve With Spiral (From Entity, Radius, Through Point)

## Adding a Floating Curve with a Spiral from the Entity End with a Specified Radius and Length to an Entity

Add a floating spiral-curve group, with a specified radius and length, to an existing entity.

When the attachment entity (1) is edited, the spiral length (3) and curve radius (2) and length (4) do not change. You can adjust or move the attachment curve and the entire curve and spiral entity moves with it.
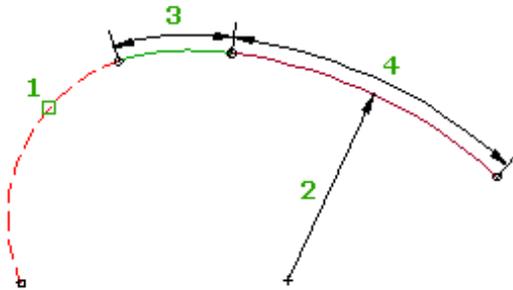


**To add a floating curve with a spiral (from entity, radius, length) off a line**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2  On the Alignment Layout Tools toolbar, click 🔂 Floating Curve With Spiral (From Entity End, Radius, Length) to add a floating curve with a spiral off a line.

   The current spiral definition is displayed at the command line.

3  Specify the line entity to add to.

4  Specify the radius.

   Specify a new value, or press Enter to accept the value that is displayed on the command line.

   ■  If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

   ■  If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5  Specify either the spiral length or the A value.

6  Specify whether the curve solution angle is either greater than or less than 180 degrees.

7  Specify an end point.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

  EditAlignment

Alignment Layout Tools Toolbar icons

 Floating Curve With Spiral (From Entity End, Radius, Length)

## Adding a Floating Reverse Curve with Spirals to a Curve

Add a floating reverse curve with spirals in situations where you need two back-to-back reverse transition spirals and the curve geometry must pass through a specified point.

Floating reverse curves are defined by either a radius and through point or two through points.

---

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

---

## Adding a Floating Reverse Curve with Spirals with a Specified Radius and Through Point to a Curve

Add a floating reverse spiral-spiral-curve group, with a specified radius and pass-through point, to a curve.

The two spirals bend in opposite directions. The curve bends in the opposite direction of the attachment curve. When the attachment curve (1) is edited, the spiral lengths (2, 4), curve radius (3) and pass-through point (5) do not change. The attachment point and the length of the reverse curve adjust to accommodate edits to the attachment curve.
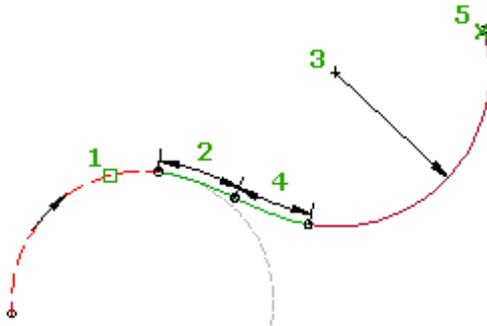


This entity does not support either of the following solutions:

■ Solutions that require a floating curve angle greater than 180 degrees. The following illustration shows both the supported and unsupported solutions:

reverse spiral-spiral, by A1/A2 or L1/L2

supported solution

not a supported solution

C

CS

CS

existing fixed or floating curve

curve by radius and pass-through point

C
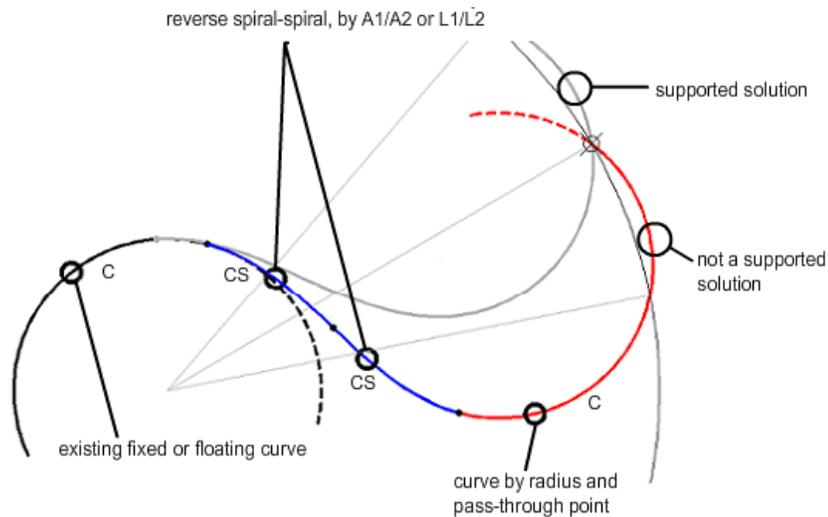
■ Compound solutions. The following illustration shows an unsupported compound solution:



compound spiral-spiral, by A1/A2 or L1/L2

curve by radius and pass-through point

CS

C

CS

C

compound solution (not supported)

existing fixed or floating curve

reverse solution (supported)

**To add a floating reverse curve with spirals (from curve, radius, through point) off a curve**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2   On the Alignment Layout Tools toolbar, click  Floating Reverse Curve With Spirals (From Curve, Radius, Through Point) to add a floating reverse curve with spirals off a curve.

The current spiral definition is displayed at the command line.

3   Specify the fixed or floating curve entity to attach to.

4   Specify either the spiral in length or the A value.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

■ If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

■ If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

5 Specify either the spiral out length or the A value.

6 Specify the radius.

7 Specify the pass through point.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖋

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

🖋 Floating Reverse Curve With Spirals (From Curve, Radius, Through Point)

## Adding a Floating Reverse Curve with Spirals with Two Through Points to a Curve

Adds a floating reverse spiral-spiral-curve group, with two specified pass-through points, to an existing curve.

The two spirals bend in opposite directions. The curve bends in the opposite direction of the attachment curve. When the attachment curve (1) is edited, the spiral lengths (2, 3) and pass-through points (4, 5) do not change. The attachment point and the radius and length of the reverse curve adjust to accommodate edits to the attachment curve.
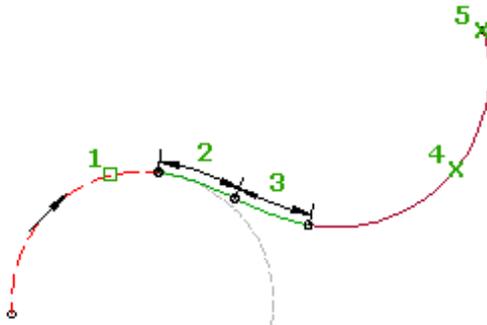


**To add a floating reverse curve with spirals (from curve, two through points) off a curve**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖋 .

2 On the Alignment Layout Tools toolbar, click 🖋 Floating Reverse Curve With Spirals (From Curve, Two Points) to add a floating reverse curve with spirals off a curve.

The current spiral definition is displayed at the command line.

**3** Specify the fixed or floating curve entity to attach to.

**4** Specify either the spiral in length or the A value.

Specify a new value, or press Enter to accept the value that is displayed on the command line.

- If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

- If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

**5** Specify either the spiral out length or the A value.

**6** Specify the pass through point.

**7** Specify the second pass through point (end point).

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

🔵 Floating Reverse Curve With Spirals (From Curve, Two Points)

## Adding Spiral-Curve-Spirals

Add a free spiral-curve-spiral group between two entities.

The type of spirals created (simple vs. compound) depends on the types of entities you attach it to. When the attachment entities (1, 2) are edited, the spiral lengths (3, 4) and curve radius (5) do not change. The attachment points and the curve length adjust to accommodate edits to the attachment entities.
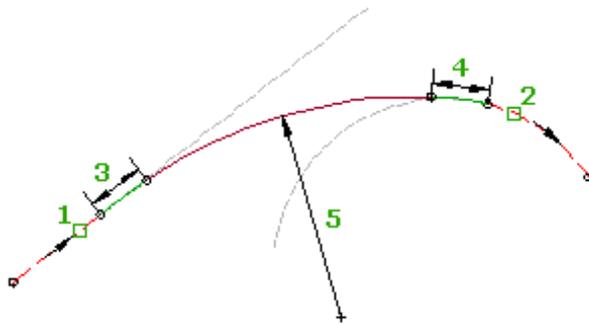
The Spiral-Curve-Spiral command is similar to the command that adds a free curve by radius between two entities. However, this command adds a transition spiral in and a transition spiral out. The curve group for this command is the same as the automatic curve group that is created when you use the Tangent-Tangent command. However, with this command you can create an SCS curve group between curves.

You cannot grip edit the spiral-curve-spiral curve group, but you can edit the entities to which the SCS group is attached. Make edits to the spiral or curve parameters in the Alignment Layout Parameters (page 1741) dialog box or the Alignment Entities (page 1735) vista.

### SCS curve group between two tangents

An SCS curve group placed between two tangents creates two simple spirals.

### SCS curve group between a curve and a tangent

An SCS curve group placed between a tangent and a curve results in the simple spiral attached to the tangent and a compound spiral attached to the curve.

### SCS curve group between two curves

An SCS curve group placed between two curves creates two compound spirals. The SCS curve group should only be placed in a configuration where the attachment to either curve forms a compound curve-curve situation.

### To add a free spiral-curve-spiral (between two entities)

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⤳ Free Spiral-Curve-Spiral (Between Two Entities).

3   Select the entity from which you want to add the spiral-curve-spiral.

4   Select the entity to which you want to add the spiral-curve-spiral.

5   Select the spiral-in length by picking two points in the drawing or by entering a spiral-in A value. Specify a new value, or press Enter to accept the value that is displayed on the command line.

   ■ If the alignment has design criteria (page 866) applied to it, the minimum value for the current design speed is displayed.

   ■ If the alignment does not have design criteria applied to it, the default value specified in the curve and spiral settings is displayed. For more information, see Curve and Spiral Settings (page 861).

6   Select the spiral-out length or enter a spiral-out A value.

7   Specify whether the curve solution angle is either greater than or less than 180 degrees.

8   Enter a radius value or select two points in the drawing to specify the radius. To display a curve preview you must either pick a point in the drawing or enter a coordinate value. Then, rubber band is drawn to the cursor from the selected point and the curve preview is displayed.

---

NOTE If you enter an A value for either the spiral-in or the spiral-out value, then the curve preview is not displayed when you specify the radius.

---

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

Free Spiral-Curve-Spiral (Between Two Entities)

## Adding Complex Free Curve Groups Between Tangents

Use the Alignment Layout Tools to add complex curve groups to your alignment. Complex curve groups consist of multiple curves and spirals and are available in compound and reverse solutions.

Complex free curve groups are defined by the radius of each curve and the length or A value of each spiral. Additionally, a second parameter must be specified for either of the two curves.

## Adding a Free Compound Spiral-Curve-Spiral-Curve-Spiral Between Two Tangents

Add a free spiral-curve-spiral-curve-spiral group between two tangents.

When the attachment tangents (1, 2) are edited, the curve radii (3, 4) and spiral lengths (5, 6, 7) do not change. The attachment points and the curve lengths adjust to accommodate edits to the attachment tangents. The length of any of the spirals can be zero.
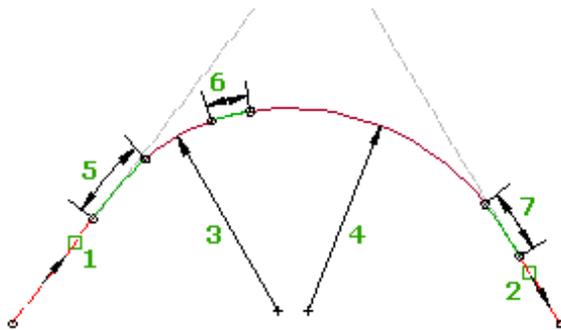


The compound Spiral-Curve-Spiral-Curve-Spiral (SCSCS) group is placed between two tangents. This group is similar to the spiral-curve-spiral (SCS) group, but it has a second curve and spiral at the end of the group.

The SCSCS group accepts zero as a spiral length, which allows you to create a multiple curve group with or without spirals. Any of the spirals in this group can have a zero length. A spiral with zero length is effectively omitted from the group. The following combinations are possible by specifying zero as the spiral length: SCSCS, SCSC, CSCS, CSC, SCCS, SCC, CCS, CC.

You cannot grip edit the SCSCS group, but you can edit the tangents to which it is attached. The point of intersection (PI) of the original tangents is maintained, which causes the entire curve group to react to

changes to either tangent. You can edit the spiral or curve parameters directly in either the Alignment Layout Parameters (page 1741) dialog box or the Alignment Entities (page 1735) vista. If a spiral in the group has zero length specified, the spiral parameters are still displayed.

**To add a free spiral-curve-spiral-curve spiral between two tangents**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2   On the Alignment Layout Tools toolbar, click ⟲ Free Compound Spiral-Curve-Spiral-Curve-Spiral (Between Two Tangents).

3   Select the tangent (the First Entity) from which you want to add the curve group.

4   Select the tangent (the Next Entity) to which you want to add the curve group.

5   Specify the length or A value of the first spiral.

---

**NOTE**  You can specify length by picking two points in the drawing. Zero is an acceptable length for any of the spirals in this curve group.

---

6   Specify the radius of the first curve.

7   Specify the length or A value of the second spiral.

8   Specify the radius of the second curve.

9   Specify the length or A value of the third spiral.

10   Enter a second parameter for one of the curves.

You can specify an extended tangent length, start point on a tangent, subtended angle, or pass-through point. Enter C to switch the curve for which you will enter the parameter.


## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icons

⟲ Free Compound Spiral-Curve-Spiral-Curve-Spiral (Between Two Tangents)


# Adding a Free Reverse Spiral-Curve-Spiral-Spiral-Curve-Spiral Between Two Tangents

Add a free reverse spiral-curve-spiral-spiral-curve-spiral group between two tangents that are nearly parallel.

This group consists of two successive spiral-curve-spiral groups in opposing directions. When the attachment tangents (1, 2) are edited, the curve radii (4, 7) and spiral lengths (3, 5, 6, 8) do not change. The attachment points and the curve lengths adjust to accommodate edits to the attachment tangents.
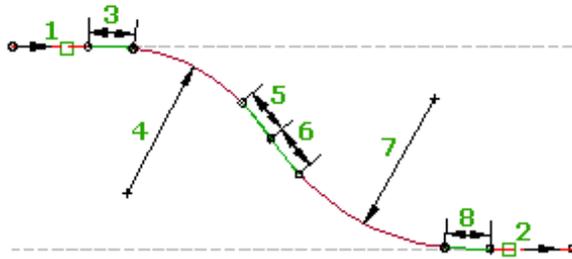
You cannot grip edit the reverse SCSSCS group, but you can edit the tangents to which it is attached. You can edit the spiral or curve parameters directly in either the Alignment Layout Parameters (page 1741) dialog box or the Alignment Entities (page 1735) vista.

**To add a free reverse spiral-curve-spiral-spiral-curve-spiral between two tangents**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏ .

2   On the Alignment Layout Tools toolbar, click 🖉 Free Reverse Spiral-Curve-Spiral-Spiral-Curve-Spiral (Between Two Tangents).

3   Select the tangent (the First Entity) from which you want to add the curve group.

4   Select the tangent (the Next Entity) to which you want to add the curve group.

5   Specify the length or A value of the first spiral (the spiral in).

   **NOTE**   You can specify length by picking two points in the drawing.

6   Specify a radius for the first curve.

7   Specify the length or A value of the second spiral (the spiral out).

8   Specify the length or A value of the third spiral (the spiral in).

9   Specify a radius for the second curve.

10   Specify the length or A value of the fourth spiral (the spiral out).

11   Enter a second parameter for one of the curves.
   You can specify a start point on a tangent, subtended angle, or pass-through point. Enter C to switch the curve for which you will enter the parameter.

## Quick Reference

Ribbon

   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏

Menu

   Alignments menu ➤ Edit Alignment Geometry

Command Line

   EditAlignment

Free Reverse Spiral-Curve-Spiral-Spiral-Curve-Spiral (Between Two Tangents)

# Alignment Labels and Tables

Use the label and table styles to control the appearance and behavior of alignment labels and tables in a drawing.

When you create alignments, they are labeled automatically with the labels specified in the Alignment Labels dialog box. The labels are applied along the alignment at specific points, for example, the major and minor stations.

With the Add Labels command, you specify the following parameters as you label the alignment:

■ Label type, such as station offset, segment, and tangent intersection

■ Label and marker styles

■ Points along the alignment

You can also use commands to insert alignment tables into a drawing. Alignment tables organize and consolidate the information about the alignment. When you insert a table into the drawing, full labels are converted to tags, and the detailed alignment information is inserted into the table.

The topics in this section describe what is unique to alignment labels and tables. For a general overview of labels, see Labels and Tags (page 1463). For an overview of tables, see Tables (page 1555).
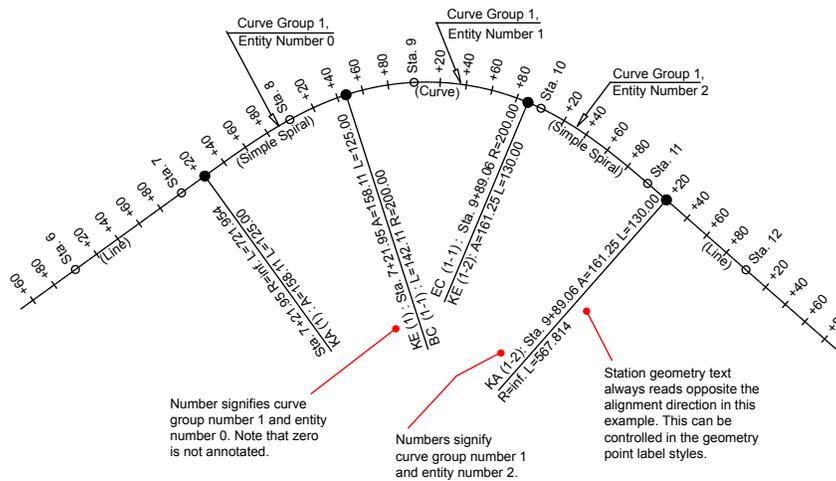


**Alignment labels**

# Alignment Label Styles

Manage alignment label styles by specifying which label styles to use when you create alignment label sets, as well as which styles to use with the Add Alignment Labels command.

In Toolspace, on the Settings tab, the Alignment collection contains the Label Styles collection. The Label Styles collection contains sub-folders for labels sets, individual types of alignment labels, and alignment table styles. You can right-click a specific type of alignment label, such as for major stations and minor stations, and either edit the default label settings or create a label style.

The types of alignment label styles that you can create and use:

| Use this type of alignment label style... | To label... |
|---|---|
| Major Station | Stations at major intervals, which use the default format of sta = <station> <units>. For example, sta = 1000 ft. |
| Minor Station | Stations at intervals that divide the major stations. To add minor station labels, you must have major station labels. |
| Geometry Point | Locations at which alignment geometry changes, which use the default format of <Geometry point>: <station>. For example, CS: 3+27+65. |
| Profile Geometry Point | Locations of profile points on the alignment geometry. |
| Station Equation | Points added on the Station Control tab of the Alignment Properties dialog box. |
| Design Speeds | Points added on the Design Speeds tab of the Alignment Properties dialog box. |
| Superelevation Critical Points | Stations at which the superelevation cross slope changes. |
| Station Offset | A point either on or adjacent to an alignment that lists station and offset information from the alignment to the point picked. |
| Line | Properties of a line entity within an alignment object. |
| Curve | Properties of a curve entity within an alignment object. |
| Spiral | Properties of a spiral entity within an alignment object. |
| Tangent Intersection | Properties of a tangent-tangent intersection (PI) and either the associated free curve or free Spiral-Curve-Spiral (SCS) group. This label type can also be applied to either individual curve or spiral entities (even within an SCS group). |

# Labeling Alignment Stations

Station labels display reference information about specific points along the length of an alignment.

AutoCAD Civil supports the following alignment station label types:

■ Major Stations

■ Minor Stations

■ Geometry Points

■ Profile Geometry Points

- Station Equations

- Design Speeds

- Superelevation Critical Points

When you create an alignment, you select a label set, which is a predefined collection of label types with associated styles. You can modify the station labels in an existing alignment, and then optionally save the changes as a label set.

## Using Alignment Label Sets

One step in creating an alignment is selecting the Alignment Label Set.

## Using Label Sets

Use the Alignment Labels dialog box to assign styles to the individual elements that you want to label on the alignment. Then save the group of labels as a label set.

For example, you can create a label set that only includes major and minor station types, and another label set that includes major and minor stations, and also geometry points, station equations, and design speeds. You can determine the style, start station, and end station for each type of individual label in the label set.

A label set can contain multiple instances of a label type. You can assign different label styles to the individual geometry points in the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types. For example, you can create a label set that contains multiple occurrences of the Geometry Points label type. One occurrence can label the Alignment Beginning and Alignment End points using a style. A second occurrence of the Geometry Points label type can label the Tangent-Tangent Intersect and Tangent-Curve Intersect points using a different style.

After you save a label set, you can reuse it by importing it to the alignment.

Label sets include the following label types:

- Major Stations

- Minor Stations

- Geometry Points

- Profile Geometry Points

- Station Equations

- Design Speeds

- Superelevation Critical Points

For more information about modifying labels, see Modifying Labels in a Drawing (page 1533). For more information about which properties of different label types can be modified, see Managing Layout Properties for Label Styles (page 1492).

**NOTE** To change the label type defaults, right-click the <Label Type> collection on the Settings tree in Toolspace and select Edit Label Style Defaults. For more information, see Default Settings for All Labels in a Label Type (page 1476).

**To add labels to a label set**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
   ◈ .

2  Select an alignment in the drawing to open the Alignment Labels dialog box.

3  In the Type list, select the type of label that you want to add.

> **NOTE** To add minor station labels, you must have at least one major station label type added to the label set.

4  In the Style list, select a style for the label type.

5  Click Add.

> **TIP** To delete a label, select the label type and click. You cannot delete a major station if it has a minor station.

6  If you select either the Geometry Points, Profile Geometry Points, or Superelevation Critical Points label type, select the specific points to label.

7  In the Style column, either click the icon to change the style or accept the label style selection.

8  In the Increment column, either enter a value to determine the increment distance between major station labels, or accept the default.

9  In the Start Station column, either clear the check box and enter a new starting station value or accept the default.

10  In the End Station column, either clear the check box and enter a new end station value or accept the default.

> **NOTE** The default start and end stations for each label type are the beginning and end of the alignment. To change either the start station or end station, clear the check box, and enter new start station and end station values.

11  In the Geometry Points To Label column, click ⊡ to update the points to label.

> **NOTE** The Geometry Points To Label column is only available for the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types.

12  Repeat Steps 2 through 10 to add more label types to the label set.

## Saving Alignment Label Sets

Create and save any number of alignment label sets to use during different phases in a project.

Copy and edit label sets from the Label Style collection in Toolspace on the Settings tab.

**To save a label set**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
   ◈ .

2  Select an alignment in the drawing to open the Alignment Labels dialog box.

3  In the Alignment Labels dialog box, determine the label types you want to save in the label set.

**4** Click Save Label Set. The Alignment Label Set dialog box is displayed.

**5** On the Information tab (page 1771), enter a name and optional description for the label set.

**6** Optionally, on the Labels tab (page 1772), change values.

## Importing Label Sets

Create and save any number of alignment label sets. Import the labels sets during different phases in a project.

**To import a label set**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
.

**2** Select an alignment in the drawing to open the Alignment Labels dialog box.

**3** In the Alignment Labels dialog box, click Import Label Set.

**4** In the Select Style Set dialog box, either select an existing label set or copy, edit, or create a label set.

## Editing Alignment Station Labels

After an alignment has been created, you can change the type and style of the station labels.

If you label an alignment created through an xref, the labels you add exist in the current drawing. If you detach the xref, labels created in the current drawing will be lost.

You can save a collection of alignment station label types and styles as a label set, which can be applied to other alignments.

**To edit alignment station labels**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
.

**2** Select an alignment in the drawing to open the Alignment Labels dialog box.

**3** In the Alignment Labels dialog box, do any of the following procedures, and then click OK.

**To add a station label type to the alignment**

**1** In the Type list, select the type of label that you want to add.

> **NOTE** To add minor station labels, you must have at least one major station label type added to the label set.

**2** In the Style list, select a style for the label type.

**3** Click Add.

**4** If you select either the Geometry Points, Profile Geometry Points, or Superelevation Critical Points label type, select the specific points to label.

**To delete a station label type from the alignment**

To delete a label, select the label type and click ✖ . You cannot delete a major station if it has a minor station.

**To change the parameters of an existing station label type**

1  In the Style column, either click the icon to change the style or accept the label style selection.

2  In the Increment column, either enter a value to determine the increment distance between major station labels, or accept the default.

3  In the Start Station column, either clear the check box and enter a new starting station value or accept the default.

4  In the End Station column, either clear the check box and enter a new end station value or accept the default.

> **NOTE** The default start and end stations for each label type are the beginning and end of the alignment. To change either the start station or end station, clear the check box, and enter new start station and end station values.

5  In the Geometry Points To Label column, click ⊡ to update the points to label.

> **NOTE** The Geometry Points To Label column is only available for the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
🏷

Menu

Click Alignments  menu ➤ Add Alignment Labels ➤ Add/Edit Station Labels.

Command Line

EditAlignmentLabels

Dialog Box

Alignment Labels Dialog Box (page 1739)
Alignment Geometry Points Dialog Box (page 1739)
Profile Geometry Points Dialog Box (page 1775)
Superelevation Critical Points Dialog Box (page 1775)

## Labeling Geometry Points

Use different styles to label specific alignment, profile, or superelevation critical geometry points along an alignment.

You can apply separate label styles to specific points in the geometry point label types. You can determine which points to label, and then apply different styles to each point type.

For example, you can create a label set that contains multiple occurrences of the Geometry Points label type. One occurrence can label the Alignment Beginning and Alignment End points using a style. A second occurrence of the Geometry Points label type can label the Tangent-Tangent Intersect and Tangent-Curve Intersect points using a different style.

Geometry point label types include:

- Geometry Points

- Profile Geometry Points

- Superelevation Critical Points

    **NOTE** Superelevation critical labels are not dynamically linked to the parent alignment. If the alignment geometry changes, the superelevation critical labels must be updated manually.

    **TIP** Superelevation critical point labels can also display a diagram that represents the cross section at each superelevation transition point. For more information, see Adding Superelevation Cross-Section Diagrams Along an Alignment (page 978).

**To add geometry point labels**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels
  .

2 Select an alignment in the drawing to open the Alignment Labels dialog box.

3 In the Alignment Labels Dialog Box (page 1739) dialog box, in the Type list, select the type of geometry point label that you want to add.

  Select either Geometry Points, Profile Geometry Points, or Superelevation Critical Points.

4 In the Style list, select a style for the label type.

  This style is used by the geometry points you specify in Step 5.

5 Click Add.

6 In the <label type> Points dialog box, select the specific points to label.

    **NOTE** If you selected Profile Geometry Points, you must specify a profile to reference in the Profile Geometry Points Dialog Box (page 1775) dialog box.

7 In the Start Station column, either clear the check box and enter a new starting station value or accept the default.

8 In the End Station column, either clear the check box and enter a new end station value or accept the default.

    **NOTE** The default start and end stations for each label type are the beginning and end of the alignment. To change either the start station or end station, clear the check box, and enter new start station and end station values.

9 Repeat Steps 1 through 7 to add more geometry point labels.

  You can add multiple instances of each label type. Each instance can label different geometry points and use a different label style and range of stations.

    **NOTE** In the Geometry Points To Label column, click  to update the points to label.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels

Menu

Click Alignmentsmenu ➤ Add Alignment Labels ➤ Add/Edit Station Labels

Command Line

EditAlignmentLabels

Dialog Box

Alignment Labels Dialog Box (page 1739)
Alignment Geometry Points Dialog Box (page 1739)
Profile Geometry Points Dialog Box (page 1775)
Superelevation Critical Points Dialog Box (page 1775)

## Adding Superelevation Cross-Section Diagrams Along an Alignment

Add block diagrams that represent the cross section at each superelevation transition point.

To add superelevation cross-section diagrams, compose a separate label style for each of the Superelevation Critical Points that you want to label. Use the Alignment Labels dialog box to add each of the point label styles to the alignment. You can save the collection of superelevation critical point labels as a label set, which you can apply to other alignments.

**To add superelevation cross-section diagrams along an alignment**

1   Compose a label style for each type of superelevation critical point that you want to label.

   Each style must contain a block component that refers to an appropriate representation of the cross section at that type of geometry point. Optionally, each style can include text, tick, and line components.

   For information about composing a label style, see Creating New Label Styles (page 1477).

2   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels .

3   Select an alignment in the drawing to open the Alignment Labels dialog box.

4   In the Alignment Labels Dialog Box (page 1739), in the Type list, select Superelevation Critical Points.

5   In the Superelevation Label Style list, select a style for the superelevation critical geometry point.

6   Click Add.

7   In the Superelevation Critical Points Dialog Box (page 1775), click  Deselect All. Select the superelevation critical points that correspond to the style you selected in Step 4. Click OK.

8   In the Start Station column, either clear the check box and enter a new starting station value, or accept the default.

9   In the End Station column, either clear the check box and enter a new end station value, or accept the default.

> **NOTE** The default start and end stations for each label type are the beginning and end of the alignment. To change either the start station or end station, clear the check box, and enter new start station and end station values.

**10** Repeat Steps 3 through 8 to add the remaining superelevation critical points.

**11** Add other alignment label types as desired.

**12** Optionally, click Save Label Set to save the current label configuration as a label set.
For more information, see Saving Alignment Label Sets (page 974).

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add/Edit Station Labels

Menu

Click Alignments menu ➤ Add Alignment Labels ➤ Add/Edit Station Labels

Command Line

EditAlignmentLabels

Dialog Box

Alignment Labels Dialog Box (page 1739)
Superelevation Critical Points Dialog Box (page 1775)

## Displaying Stations in Station Index Format

Use the Station Index Format to label a station value as a count that is relative to a distance.

This distance is referred to as a station index increment. Specify the station index increment in the Alignment Labels dialog box. Format the station index label using the Label Style Composer Dialog Box (page 1942).

**To display stations in station index format**

**1** In Toolspace, on the Settings tab, right-click a station label type. Click either Edit or New.

**2** In the Label Style Composer, click the Layout tab.

**3** Under Text, click ⋯ in the Contents Value column.

**4** In the Text Component Editor, click the Format Value column. Select Station Index Format from the list.

**5** Specify other modifiers as necessary.

**6** Click ⇨ to insert the property into the Text Component Editor window.

**7** Click OK.

## Changing the Station Character Display

Use the Station Character setting to change the display of the character in station labels.

The following station character formats are available:

- 10+50.00
- 10-50.00
- 10_50.00
- 1050.00
- 10/50.00
- 10C50.00
- 10K50.00

**To change the display of the station character**

1  In Toolspace, on the Settings tab, right-click a station label type. Click either Edit or New.

2  In the Label Style Composer, click the Layout tab.

3  Under Text, click ⬚ in the Contents Value column.

4  In the Text Component Editor, in the preview pane, select the station value property formula.

5  In the Properties list, select Station Value.

6  Click the Station Character Value column. Select the desired character from the list.

7  Specify other modifiers as necessary.

8  Click ⇨ to insert the property into the Text Component Editor window.

9  Click OK.

# Adding Descriptive Geometry Point Text

Use the Abbreviations tab in the Drawing Settings dialog box to compose labels that include geometric property information about the entities that come before and after geometry points.

**See also:**

- Abbreviations Tab (Drawing Settings Dialog Box) (page 1855)

**To add descriptive geometry point text**

1  In Toolspace, on the Settings tab, right-click the drawing and click Edit Drawing Settings. Click the Abbreviations Tab (Drawing Settings Dialog Box) (page 1855).

2  In the Alignment Geometry Point Entity Data section, select the entity endpoint type. Click ⬚ in the Value column to modify the label for that type.

3  In the Text Component Editor, select a property from the list and select modifiers as necessary.

4  Click ⇨ to insert the property into the Text Component Editor window.

5  Insert other properties and modifiers as necessary. Click OK.

6  Repeat Steps 2-5 to format other alignment endpoint entities listed on the Abbreviations tab.

**7** In Toolspace, on the Settings tab, right-click a <geometry point label>. Click either Edit or New.

**8** In the Label Style Composer, click the Layout tab.

**9** Under Text, click [...] in the Contents Value column.

**10** In the Properties list, select Geometry Point Entity Before Data. Click ⇨ .

**11** In the Properties list, select Geometry Point Entity After Data. Click ⇨ .

**12** Click OK.

The text you compose on the Abbreviations tab for the alignment endpoint entities is included in the labels that display before and after the geometry point.

# Labeling Alignment Line, Curve, and Spiral Entities

You can label components along an alignment by using specified label styles.

To place the label, specify the alignment and the component in the drawing, such as a line, curve, or spiral segment. Select either a single segment or intersection or label the whole alignment using the multiple options.

For more information about modifying labels, see Modifying Labels in a Drawing (page 1533). For more information about modifying label properties, see Managing Layout Properties for Label Styles (page 1492).

## Adding Labels to Single Segments in Alignments

Add single segment labels to the lines, curves, or spirals with Line, Curve, and Spiral label styles.

Labels are placed on each line (1), curve (2) or spiral entity that you select.
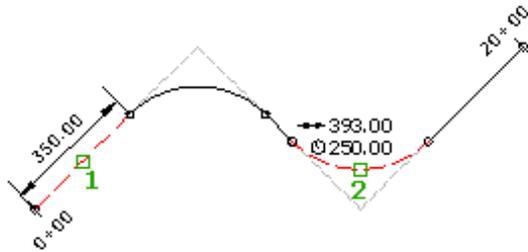


**To label alignment line segments**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Single Segment ✎ .

**2** Select a location on the alignment line or segment to label.

**3** In the Label Type list, select Single Segment.

**4** Specify label styles for Line, Curve, and Spiral.

**5** Click Add. In the drawing, select the alignment.

**6** In the drawing, select the segments you want to label.

**Quick Reference**

## Adding Multiple Segment Labels to Alignments

Automatically label all lines, curves, and spirals in the alignment with Line, Curve, and Spiral label styles.

Labels are placed on each line, curve, and spiral entity in the selected alignment.
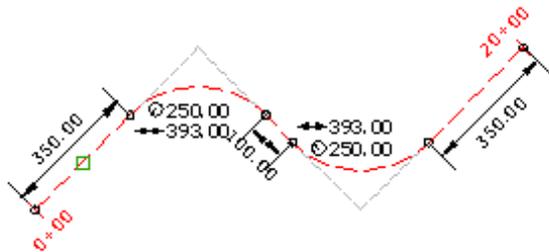


**To label multiple alignment line segments**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Add Alignment Labels ✎ .

2  In the Add Labels dialog box, verify that Alignments is displayed in the Feature list.

3  In the Label Type list, select Multiple Segment.

4  Specify label styles for Line, Curve, and Spiral.

5  Click Add. In the drawing, select the alignment.
   Labels are displayed on all lines, curves, and spirals in the alignment.

**Quick Reference**

Command Line

AddAlignLabels

# Labeling Station Offset and Tangent Intersection Points in Alignments

You can label a station offset or a tangent intersection with specified offset and marker label styles.

You can label individual tangent-intersection points or use the multiple tangent intersection command and label all points. The multiple tangent intersection command labels the PI points of all intersecting tangents, along with the PI point for any curves, or spiral-curve-spiral groups.

**NOTE** You cannot label fixed spirals with the multiple tangent intersection label command.

## Adding Station Offset Labels - Relative Point

Add station offset labels at selected points along the alignment. You must also specify a style for the point marker.

For the selected alignment (1), you specify the desired station (2) and offset distance (3). If the alignment is moved, the label moves with it and maintains the relative station and offset location.
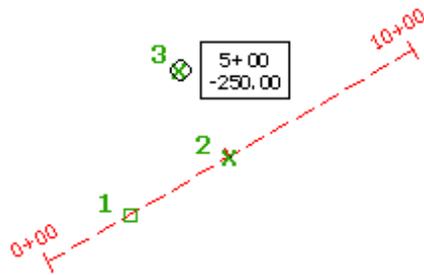


**NOTE** If you move the alignment, labels that you add using the station offset labels command move with the associated point.

**To add station offset labels for relative points**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Station/Offset  .

2   Select the station and offset distance to label.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Station/Offset 

Menu

Click Alignments menu ➤ Add Alignment Labels ➤ Station/Offset

Command Line

AddAlignOffLbl

## Adding Station Offset Labels - Fixed Point

Add station offset labels at fixed points in the drawing. Because the Station Offset label uses a point marker to locate the station offset point, you must also specify a Marker Style.

For the selected alignment (1), you specify the XY location of the desired point (2). If the alignment is moved, the point's X-Y location is maintained, but the labels update to show new station and offset distances.
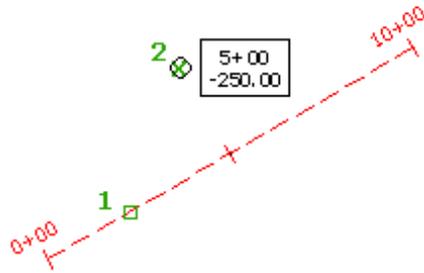


**NOTE** If you edit the alignment, labels that you add using the Station/Offset - Fixed Point command do not move.

**To add station offset labels at a fixed point**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Station/Offset – Fixed ✎ .

2  Select the point location to label.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Station/Offset – Fixed ✎

Menu

Click Alignments menu ➤ Add Alignment Labels ➤ Station/Offset - Fixed Point

Command Line

AddAlignOffXYLbl

## Adding Tangent Intersection Labels

Add tangent intersection labels to selected intersections on the alignment.

You can label the intersection point, or label the properties of a curve or a spiral-curve-spiral group at the tangent intersection.

---
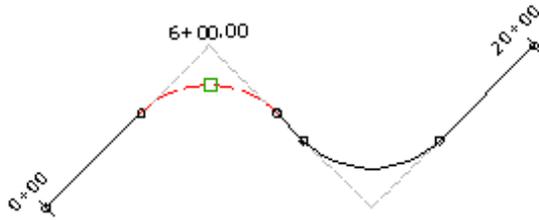
**NOTE** Complex free curve groups (compound SCSCS and reverse SCSSCS) do not support tangent intersection labels. For more information on complex curve groups, see Adding Complex Free Curve Groups Between Tangents (page 968).

---

**To add tangent intersection labels**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Tangent Intersection 🏷 .

2 Select the tangent intersection to label.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Tangent Intersection 🏷

Menu

Click Alignments menu ➤ Add Alignment Labels ➤ Tangent Intersection

Command Line

AddAlignTangentIntLbl

## Adding Multiple Tangent Intersection Labels

Add multiple tangent intersection labels at every point of intersection (PI) on the alignment, including the actual point of intersection of two tangents, as well as the virtual locations of the PI for curves and spiral-curve-spiral groups.

Labels are placed at all tangent intersection points for the selected alignment.
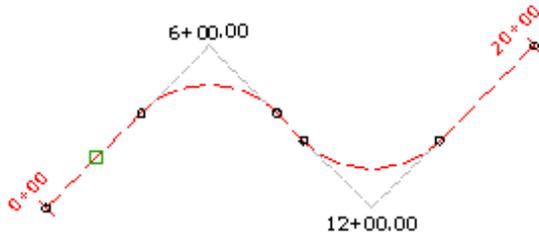
**NOTE** Complex free curve groups (compound SCSCS and reverse SCSSCS) do not support PI labels. For more information on complex curve groups, see Adding Complex Free Curve Groups Between Tangents (page 968).

**To add multiple tangent intersection labels**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Multiple Tangent Intersection 🏷 .

2   Select the tangent intersections to label.

### Quick Reference

Ribbon

  Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Alignment ➤ Multiple Tangent Intersection 🏷

Menu

  Click Alignments menu ➤ Add Alignment Labels ➤ Multiple Tangent Intersection

Command Line

  AddAlignTangentIntLbls

# Alignment Table Styles

Manage alignment table styles, which define the type of information that is placed in the alignment table on the Settings tab.

The types of alignment table styles that you can create and use:

| This Alignment table style... | Places the following alignment information in the table... |
| --- | --- |
| Line | Geometric information for points on any solved line portion of the alignment |
| Curve | Geometric information for points on any solved curve portion of the alignment |
| Spiral | Geometric information for points on any solved spiral portion of the alignment |

| This Alignment table style... | Places the following alignment information in the table... |
|---|---|
| Segment | Combines geometric information for lines, curves, and spirals into one table |

# Adding Alignment Tables

Create alignment tables that contain information about the alignment components, such as lines, curves, spirals, and segments.

Alignment segments must be labeled before you add a table to the drawing. When you add the table, the labels are converted to tags.

Use alignment tables to present alignment data in a concise format and to reduce the labels to compact tags that reference the table rows.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. The procedures in this topic explain how to access the alignment table creation commands, and provide links to reference information about the Table Creation dialog box. For information about modifying tables, see Modifying Tables (page 1567).

**To create a line table**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Line ⊞ .

2  In the Table Creation (page 2441) dialog box, select a Table Style and a Table Layer.

3  Specify the label styles or select labels in the drawing to create a tag child style for the labels.

4  Select the Apply check box.

5  Select the Split Table check box and enter values.

6  Under Behavior, select either Static or Dynamic.

7  Click the upper left corner in the drawing where you want the table to display.

**To create a curve table**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Curve ⊞ .

2  In the Table Creation (page 2441) dialog box, select a Table Style and a Table Layer.

3  Specify the label styles or select labels in the drawing to create a tag child style for the labels.

4  Select the Apply check box.

5  Select the Split Table check box and enter values.

6  Under Behavior, select either Static or Dynamic.

7  Click the upper left corner in the drawing where you want the table to display.

**To create a spiral table**

1  Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Spiral.

2  In the Table Creation (page 2441) dialog box, select a Table Style and a Table Layer.

3  Specify the label styles or select labels in the drawing to create a tag child style for the labels.

**4** Select the Apply check box.

**5** Select the Split Table check box and enter values.

**6** Under Behavior, select either Static or Dynamic.

**7** Click the upper left corner in the drawing where you want the table to display.

**To create a segment table**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Segment ⊞ .

**2** In the Table Creation (page 2441) dialog box, select a Table Style and a Table Layer.

**3** Specify the label styles or select labels in the drawing to create a tag child style for the labels.

**4** Select the Apply check box.

**5** Select the Split Table check box and enter values.

**6** Under Behavior, select either Static or Dynamic.

**7** Click the upper left corner in the drawing where you want the table to display.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Line ⊞

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Curve ⊞

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Spiral ⊞

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Alignment ➤ Add Segment ⊞

Menu

Click Alignmentsmenu ➤ Add Tables ➤ Add Line
Click Alignmentsmenu ➤ Add Tables ➤ Add Curve
Click Alignmentsmenu ➤ Add Tables ➤ Add Spiral
Click Alignmentsmenu ➤ Add Tables ➤ Add Segments

Command Line

AddAlignmentLineTable
AddAlignmentCurveTable
AddAlignmentSpiralTable
AddAlignmentSegmentTable

Dialog Box

Table Creation Dialog Box (page 2441)

# Editing Alignments

Edit an alignment by modifying the geometry or parameter values.

You can grip edit an alignment, modify geometric values, or add constraint-based line, curve, and spiral entities.
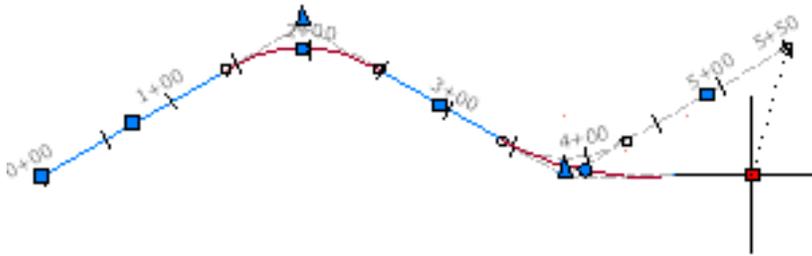
You use the Alignment Layout Tools toolbar, Alignment Layout Parameters dialog box, and Alignment Entities vista to edit an alignment. These dialog boxes remain open if you either grip edit or deselect the alignment, or enter another command.

If you select an alignment for editing and then select a second alignment, the second alignment becomes the active alignment. The most recently selected alignment has the focus of the Alignment Layout Tools toolbar and the Alignment Entities vista. The last object selected becomes the current object and has focus of the alignment editing tools.

**See also:**

■　Alignment Properties (page 863)

## Selecting Alignment Entities

Select an entire alignment, a single sub-entity, or a range of sub-entities.

When a single sub-entity is selected, the Alignment Layout Parameters dialog box displays the parameters of the selected entity.

After you have selected an alignment entity (sub-entity), you can edit it using various tools.

If the Alignment Entities  (page 1735) vista and Alignment Layout Tools (page 1747) toolbar is open, selecting a sub-entity or range of sub-entities in an alignment causes the selected sub-entities to display in the Alignment Entities vista. Selecting sub-entities or ranges of sub-entities in a different alignment causes that range of selected sub-entities to display in the Alignment Entities vista and have focus of the Alignment Layout Tools toolbar. The last object selected becomes the current object and has focus of the alignment editing tools.

**To select an entire alignment**

1　Click an alignment in the drawing.

　　Grips appear along the alignment.

2　Use the desired tools to edit the alignment:

| To Edit the alignment... | see this topic: |
| --- | --- |
| by adding constraint-based sub-entities | Editing Alignment Points of Intersection (page 992) |

| To Edit the alignment... | see this topic: |
|---|---|
| by entering precise attribute values | Editing Alignment Entity Layout Parameters (page 994) |
| using grips | Editing Alignments Using Grips (page 996) |

**To select a single alignment sub-entity**

1   Ctrl+click an alignment sub-entity in the drawing.

2   Edit the alignment sub-entity by either manipulating the grips (page 996) or entering specific attribute values (page 994).

**NOTE** The Alignment Entities Vista (page 1735) displays only the selected sub-entity attributes.

**To select a range of alignment sub-entities**

1   In the drawing window, Ctrl+click the first alignment sub-entity in the desired range.

2   Ctrl+click the last alignment sub-entity in the range.

3   Right-click one of the selected sub-entities and select Edit Alignment Geometry.

4   Click  Alignment Grid View on the Alignment Layout toolbar (page 1747).

5   Edit the alignment sub-entities by either manipulating the grips (page 996) or entering specific attribute values (page 994)

**NOTE** The Alignment Entities vista displays only the attributes of the sub-entities in the range.

## Quick Reference

Ribbon

   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 

Menu

   Alignments menu ➤ Edit Alignment Geometry

Command Line

   EditAlignment

Dialog Box

   Alignment Layout Tools (page 1747)
   Alignment Entities Vista (page 1735)
   Alignment Layout Parameters Window (page 1741)

# Converting an AutoCAD Line or Arc to an Alignment Sub-Entity

Create a fixed two-point line or three-point curve alignment entity from an AutoCAD line or arc.

The new sub-entity becomes part of the solved alignment geometry if either of the following two conditions are met:

■ Before it is converted, it must be attached to an unattached end point of another solved entity in the alignment.

■ After it has been converted, it can be joined to the solved alignment geometry using the alignment layout tools.
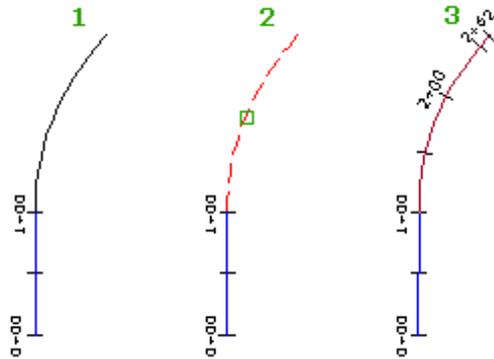


This command is useful when you must add an AutoCAD line or arc onto an existing alignment. AutoCAD lines and arcs can be created using the best fit entities or other lines and curves commands. See Lines and Curves (page 559) for more information.

**To convert an AutoCAD line or arc to an alignment sub-entity**

1 Create an alignment or find an existing one.

2 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉 .

3 On the Alignment Layout Tools toolbar, click Convert AutoCAD Line And Arc.

4 Select the AutoCAD entity.

5 If necessary, reverse the direction of the alignment sub-entity. See Reversing Alignment Sub-Entity Direction (page 1012) for more information.

NOTE If the sub-entity you converted is hidden behind the source AutoCAD entity, use Shift+spacebar to select it. Hover the cursor over the white AutoCAD entity. Press and hold Shift, then press the spacebar to highlight the alignment sub-entity. Click to select the alignment sub-entity, then release the Shift key. Click the alignment sub-entity again to select it and continue with the command.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

Convert AutoCAD Line And Arc

# Editing Alignment Points of Intersection

Edit an alignment geometry by working with the points of intersection (PI).

You can access the PI commands on the Alignment Layout Tools (page 1747) toolbar.

## Adding a Point of Intersection to an Alignment

Break a fixed line into two adjacent fixed lines by creating a point of intersection (PI) at a specified point.

When you select a point (1), the new PI (2) is created. At the new PI, endpoints are created for the new lines, which maintain tangency to one another.
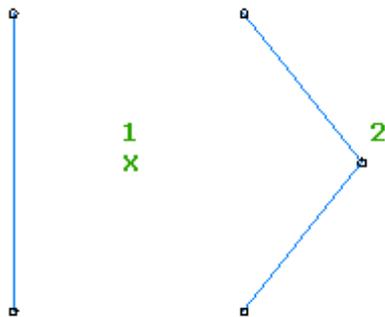


**To add a PI to an alignment**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor.

2   On the Alignment Layout Tools toolbar, click ⛰ Insert PI.

3   In the drawing window, click the location for the new PI.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

⛰ Insert PI

## Deleting a Point of Intersection from an Alignment

Create a single tangent from two adjacent tangents by removing their point of intersection (PI).

When the PI (1) is deleted (2), the tangent endpoints are removed.



**To delete a PI from an alignment**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉 .

2   On the Alignment Layout Tools toolbar, click △✗ Delete PI.

3   In the drawing window, click a point near the PI that you want to remove.

## Quick Reference

Ribbon

   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 🖉

Menu

   Alignments menu ➤ Edit Alignment Geometry

Command Line

   EditAlignment

Alignment Layout Tools Toolbar icon

   △✗ Delete PI

## Breaking Apart an Alignment Point of Intersection

Separate the point of intersection (PI) where the endpoints of two fixed or floating tangents meet.

When you break a PI (1), the tangent endpoints are separated by a specified distance (2). When a PI is broken (3), the second tangent is no longer part of the solved alignment geometry.

**To break apart an alignment PI**

**1** Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

**2** On the Alignment Layout Tools toolbar, click ⌁ Break-Apart PI.

**3** In the drawing window, click a point near the PI that you want to break apart.

**4** Specify the distance that you want to break apart the PI.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

⌁ Break-Apart PI

## Editing Alignment Entity Layout Parameters

Use the Alignment Entities vista and Alignment Layout Parameters dialog box to display and edit parameters of alignment entities that are part of the solved alignment geometry.

The Alignment Entities vista displays a form that is like a spread sheet, where each row represents an alignment entity and each column represents a parameter. You can right-click a column heading to display options for customizing the display of detail in the column views.

You can use the alignment grid view, which is also known as the Alignment Entities vista, to quickly scroll through all the sub-entities that make up the alignment. You can select a sub-entity in the Alignment Entities vista, and then edit it in the Alignment Layout Parameters dialog box. Attributes in the Alignment Entities vista and Alignment Layout Parameters window are automatically updated as the alignment is edited.

You can use the Alignment Layout Parameters dialog box to display and edit the many attributes for a single alignment sub-entity. The parameters that are displayed are dependent upon the constraints you used to create the entity.

The Alignment Layout Parameters dialog box contains up to three panels. The Layout Parameters panel, which displays the geometric parameters of the selected sub-entity, is always present. In this panel, the Value column lists the actual value of each sub-entity Parameter. If the alignment was created using the criteria-based design (page 2471) feature, the Constraint column lists the value to which a Parameter is constrained by the specified design criteria. Use the Value and Constraint columns together to view whether the sub-entity parameters meet the specified design criteria. If the alignment was not created using the criteria-based design feature, the Constraint column is not available.

Two additional panels may be present on the Alignment Layout Parameters dialog box if the alignment has design criteria applied to it. If the alignment has a design criteria file (page 2472) associated with it, the criteria from the file are displayed in the Design Criteria panel. If the alignment has a design check set (page 2472) associated with it, the name and contents of the design check set are displayed in the Design Checks panel.

If any of the parameters violate either a design check or the standards specified in the design criteria file, a ⚠ warning symbol appears next to the violation. For more information, see Viewing and Correcting Alignment Design Criteria Violations (page 1008).

**NOTE** The attributes of a curve or other alignment entity that you can edit depends on the method you used to create the entity. Generally speaking, you can edit the numeric value of a parameter if the parameter was used to define the entity. For all other parameters, numeric values are calculated and displayed but cannot be edited.

**See also:**

■  Editing Best Fit Alignment Entities (page 1003)

**To edit alignment layout parameters using the Alignment Entities vista and Alignment Layout Parameters dialog box**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝 .

2  On the Alignment Layout Tools toolbar, click ▦ Alignment Grid View.

   The Alignment Entities Vista (page 1735) displays the attributes of only the selected alignment entities. If you select another entity using Ctrl+click, it is added to the list.

3  On the Alignment Layout Tools toolbar, click ▦ Sub-Entity Editor.

   The Alignment Layout Parameters Window (page 1741) displays detailed attributes of a single alignment entity that is highlighted on the Alignment Entities vista. Selecting another entity in the Alignment Entities vista causes the new entity's attributes to display in the Alignment Layout Parameters dialog box.

   **NOTE** You can edit only values that are available. You cannot edit unavailable (shaded) text in either the Alignment Entities vista or Alignment Layout Parameters dialog box.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 📝

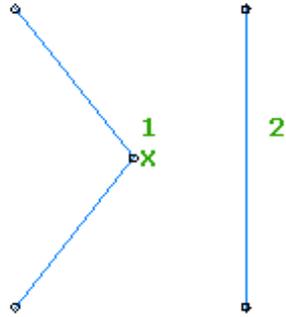Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

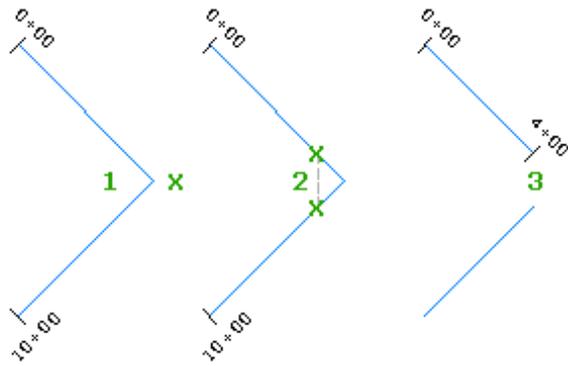Alignment Layout Tools Toolbar Icons

⊞ Alignment Grid View

📝 Sub-Entity Editor

Dialog Box

# Editing Alignments Using Grips

Use grips to change the vertical curves and tangents in an alignment graphically.

Use editing grips to relocate or modify curves and tangents using visual design cues. If you created alignment entities using the constraint-based tools, some editing grips are not available.

Grips are oriented in the direction of the WCS (World Coordinate System) in which the object currently is displayed. The exception is constrained grips, which indicate a particular constraint direction.

| Use this grip... | To do this... |
| --- | --- |
| ■<br><br>AT END OF LINE OR CURVE | Move an unconstrained pass-through point on a line or curve. On a line, moving this grip affects the length and angle of the line. On a curve, moving this grip does not affect the radius of the curve to which it belongs, but it can affect the radius of another attached entity. |
| ■<br><br>AT CENTER OF LINE | Move an entire line. Moving this grip only affects the line location. |
| ▶<br><br>AT END OF LINE OR CURVE | Change the constrained length of a line or curve. This grip appears on lines and curves that have a directly editable length and is aligned with the vector at the end of the line or curve. Moving this grip does not affect the radius of a curve or angle of a line. |
| ●<br><br>AT CURVE CENTER POINT | Change the center point of a circle. Moving a center point changes only the center point. |
| ◆<br><br>AT PASS-THROUGH POINT | Change the pass-through point and radius of a curve or circle. Moving a pass-through point affects the radius of the curve. |
| ▲<br><br>AT POINT OF INTERSECTION | Change where two tangent points meet. This grip is oriented up with the top pointing toward the Y axis of the world coordinate system. |

| Use this grip... | To do this... |
|---|---|
| <br>AT PASS-THROUGH POINT | Change the radius or through point of the curve. The triangle grip appears on curves that have a directly editable radius and is oriented in and constrained to the direction of the change. The circle grip changes the pass-through point and radius of the curve. |

When a grip is selected, you can use either dynamic input or the Alignment Entities (page 1735) vista and Alignment Layout Parameters (page 1741) dialog box to enter precise attribute values for the grip.

**To use grips to edit a fixed line**

1 In the drawing, click an alignment containing a fixed line.

Grips for a fixed line with two pass-through points:
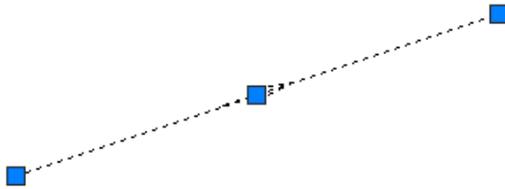


2 Use the grips to edit the line:

- Use the end grips to adjust the line length and pass-through point.

- Use the center grip to move the location of the line.

**To use grips to edit a fixed curve with three pass-through points**

1 In the drawing, click an alignment containing a fixed curve through three points.

Grips for a fixed curve with three pass-through points:
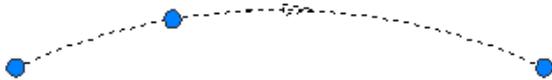


2 Move any grip to change a pass-through point, which changes the radius of the curve.

**To use grips to edit a fixed curve with a center point and radius**

1 In the drawing, click an alignment that contains a fixed curve with a center point and radius.

Grips on a fixed curve with a center point and radius:
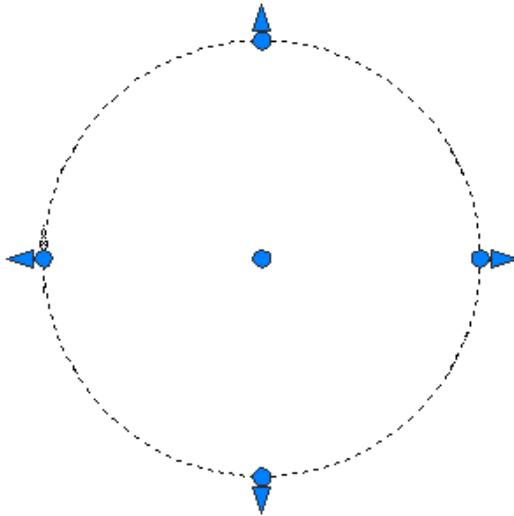
**2** Use the grips to edit the curve:

■ Move a triangular grip to change the radius.

■ Move the center grip to change the center point of the circle.

■ Move a circular grip on the circle to change a pass-through point.

**To use grips to edit a fixed curve with a center point and a pass-through point**

**1** In the drawing, click an alignment that contains a fixed curve with a center point and a pass-through point.

Grips for a fixed curve with a center point and a pass-through point:
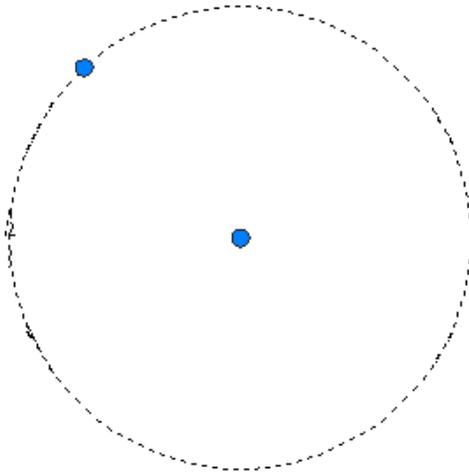


**2** Use the grips to edit the curve:

■ Move the center grip to change the center point of the circle.

■ Move the grip on the circle to change the pass-through point.

**To use grips to edit fixed or floating intersecting lines**

**1** In the drawing, click an alignment that contains either a fixed or a floating line.

PI (point of intersection) grip where the endpoints of a fixed or floating line meet:
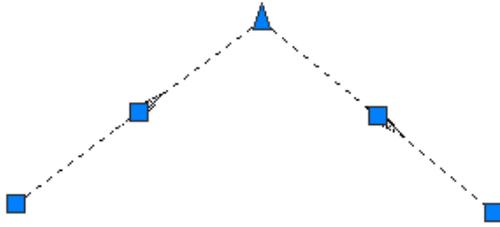
**2** Move the PI grip in any direction to edit the shared endpoints of the two lines.

**To use grips to edit a floating line with a pass-through point**

**1** In the drawing, click an alignment that contains a floating line with a pass-through point.
Grip for a floating line with a pass-through point:
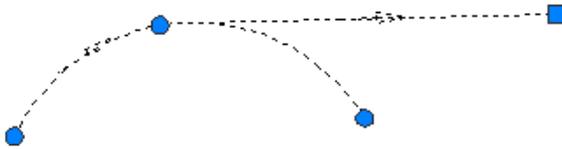
**2** Move the square grip to edit the endpoint of the line.

**To use grips to edit a floating line attached to an entity end**

**1** In the drawing, click an alignment that contains a floating line attached to an entity.
Grips for a floating line attached to an entity:
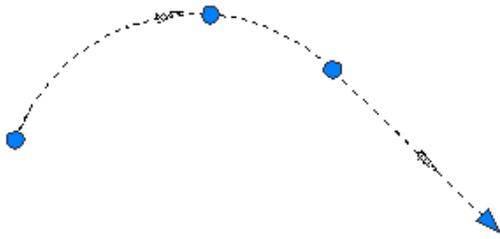
**2** Move the triangular grip to change the length of the line.

**NOTE** You cannot move the triangular grip to the beginning of the line grip. Zero length lines are not permitted.

**To use grips to edit a floating curve with a pass-through point and radius**

**1** In the drawing, click an alignment that has a floating curve with a pass-through point and radius.
Grips for a floating curve with a pass-through point:
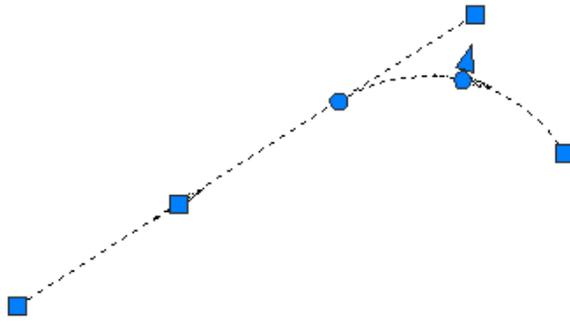
**2** Use the grips to edit the curve.

- Moving the first circular grip, located at the PC/PT point of the curve, relocates the point of tangency (PT). Use this grip to create a compound or reverse curve with another curve attached to the same entity.
  Changing a curve using the PC/PT grip:
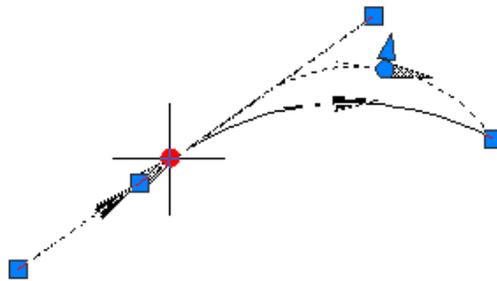
- Moving the second circular grip, located at the mid-point of the curve, modifies the radius while changing the pass-through point.
  Changing a curve using the midpoint grip:
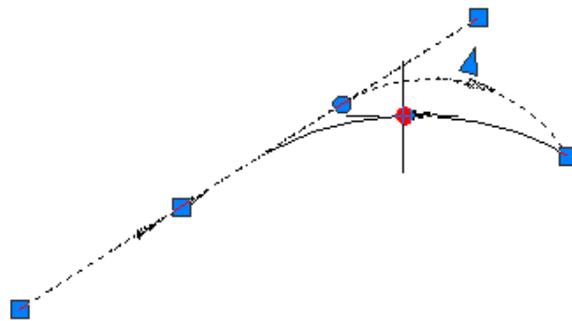
- Moving the square grip at the end of the curve changes the pass-through point of the curve while holding the curve radius.
  Changing a curve using the end grip:
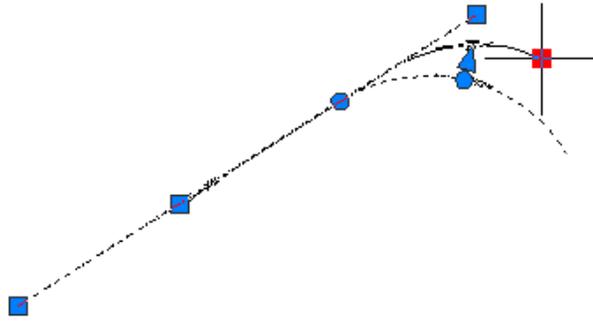
- Moving the triangular grip changes the radius while holding the pass-through point.
  Changing a curve using the radius grip:
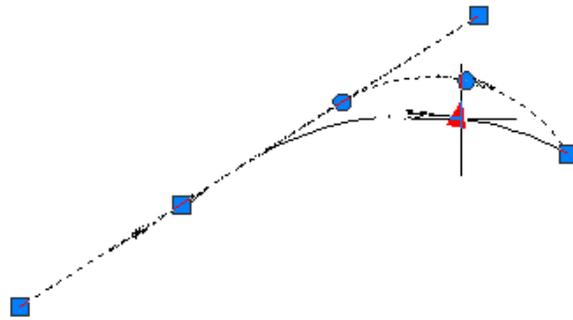


**To use grips to edit a floating curve attached to an entity end with a pass-through point**

1   In the drawing, click an alignment that contains a floating curve attached to an entity end with a pass-through point.

   Grips for a floating curve attached to an entity end with a pass-through point:
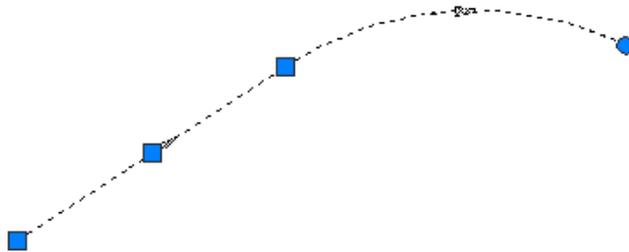


2   Move the circular grip to change the pass-through point at the end of the curve. Moving the pass-through point also changes the radius.

**To use grips to edit a free curve by radius**

1   In the drawing, click an alignment that has a free curve. Move any grip to modify the curve radius.

   Grips for changing the curve radius:
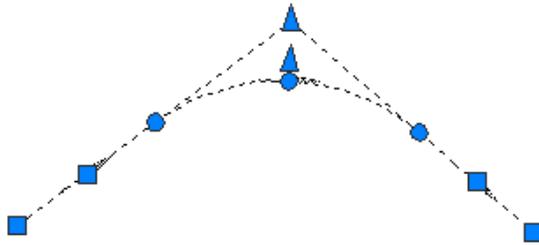
**2** Use the grips to edit the curve:

- Move the circular grips along tangent entities to adjacent PC/PT points on the opposite end of the same tangent to create reverse or compound solutions.
  Endpoint grips for dragging the beginning or end of the curve along the attachment entity:
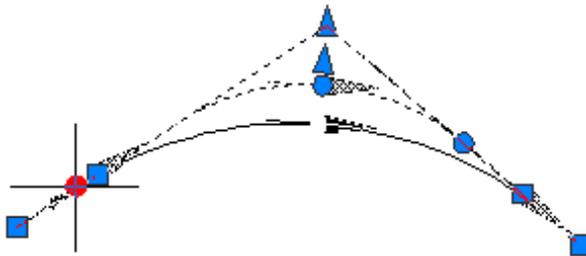
- Move the circular, mid-point grip to modify the pass-through point and radius for the curve. Move the triangular grip to modify the radius for the curve while holding the pass-through point. By moving either of these grips, you can create a curve of less than or greater than 180 degrees. Changing the radius to less than 180 degrees:
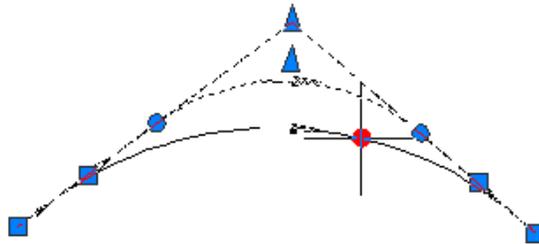
Changing the radius to greater than 180 degrees using the circular pass-through point grip:
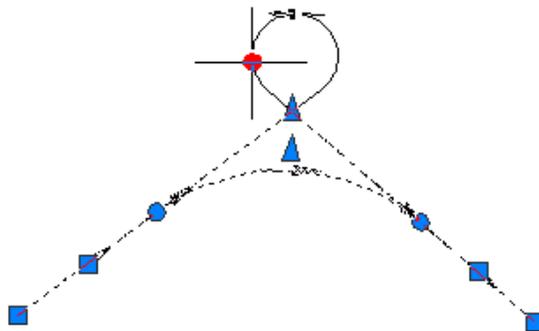
Changing the radius to greater than 180 degrees using the triangular radius grip:
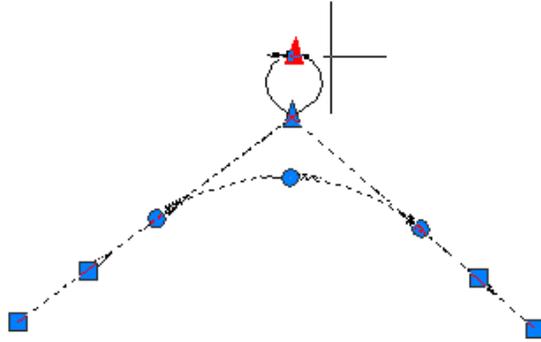
**To use grips to edit a free curve with a pass-through point**

**1** In the drawing, click an alignment with a free curve and a pass-through point.

Grips for a free curve with a pass-through point:
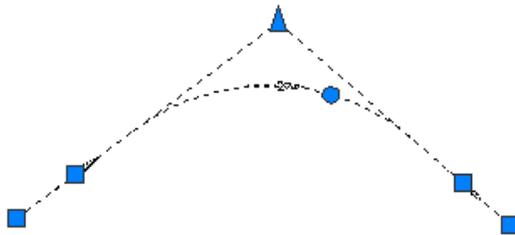


**2** Move the circular grip to change the pass-through point of the curve. This pass-through point is maintained when you edit the attached entities.

# Editing Best Fit Alignment Entities

Modify the regression data for alignment entities that were created by best fit.

The regression data can be modified, and then applied to the entity. If the entity is grip edited, it can be resynchronized with the original regression data.

The Regression Data vista displays the source regression data from which the best fit entity was created. The regression data can be modified either before or after the entity has been created. The information in the Regression Data vista can be changed, and then applied to the entity. If the entity parameters are changed outside the Regression Data vista, the entity can be synchronized to the current regression data.

The Alignment Entities vista displays a form that is like a spread sheet, where each row represents an alignment entity and each column represents a parameter. When an entity is created by best fit, Best Fit is displayed

in the Constraint2 column. Click  in this column to edit the source regression data.

Regression data is not dynamically linked to the best fit entity. If the entity is edited,  appears in both the Regression Data and Alignment Entities vistas.

**NOTE** You can edit only values that are available, you cannot edit unavailable (shaded) text.

**Editing Floating Best Fit Entities**

The manner in which a best fit floating entity reacts to edits to the attachment entity depends on whether the best fit entity is synchronized with the best fit sample data:

■ If the floating best fit entity is synchronized with the best fit sample data, it will maintain tangency when the attachment entity is edited. The floating best fit entity geometry will change to adapt to changes to the attachment entity, and continue to honor the regression points.

■ If the floating best fit entity is not synchronized with the best fit sample data, it will behave as a normal floating entity. The floating best fit entity will remain tangent to the attachment entity, and its geometry will not change to honor the regression points.

**To edit best fit regression data**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2   On the Alignment Layout Tools toolbar, click  Edit Best Fit Data For All Entities.

> **NOTE** You can also open the Regression Data vista from the Alignment Entities vista. In the Constraint2 column, click  .

3   In the Regression Data vista, edit the regression data as needed.
    You can add, edit, or remove regression points.

4   Click  to apply the changes and close the Regression Data vista.

**To synchronize a best fit alignment entity to the regression data**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2   On the Alignment Layout Tools toolbar, click  Edit Best Fit Data For All Entities.

3   In the Regression Data vista, click  Synchronize Entity To Best Fit Sample Data.

> **NOTE** You can also synchronize the entity to the regression data from the Alignment Entities vista. In the Constraint2 column, click  .

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor 

Menu

Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar Icon

Edit Best Fit Data For All Entities

Regression Data Vista Icon

Synchronize Entity To Best Fit Sample Data

Alignment Entities Vista Icon



Dialog Box

Regression Data Vista (for Best Fit Entity Analysis) (page 1990)
Alignment Entities Vista (page 1735)

# Editing Offset Alignments and Widenings

You can edit the parameters of an offset alignment or widening, split it into sections, and attach additional widenings.

Two editing methods are available:

■   Edit values in the Offset Parameters dialog box

■   Grip editing

### Editing Offset Parameters

This editing method offers precise control of the dimensions and shape of an offset alignment or widening. Parameters for each region of the object are displayed in a tabular format in the Offset Parameters (page 1773) dialog box.

When you click on the top heading for a region, the region is highlighted in the drawing.
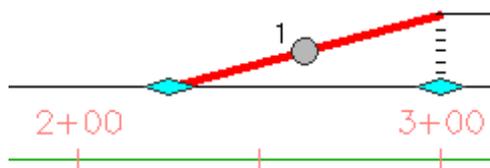


**Example of a highlighted region**

### Grip Editing

This editing method enables you to use freehand controls to reshape an offset or widening. Dimensions are also available in tooltips.
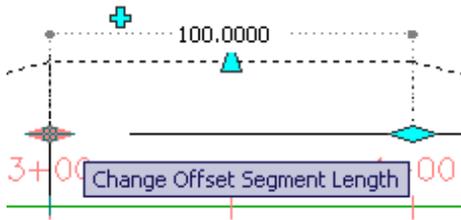
**Examples of tooltips for grip editing**

The various grips and their use are described in the following table.

| Grip | Function |
|---|---|
|  | Adjusts the width of the widening. |
|  | Adjusts the length of a region along the alignment. |
|  | Adds a widening by splitting the current offset or widening into two regions of equal length. |
|  | On a transition, switches the available grip set from transition grips to widening grips. |
|  | Adjusts the length of a transition. |

**To edit offset parameters**

1  Click the offset alignment. Click Offset Alignment tab ➤ Modify panel ➤ Offset Parameters .

2  In theOffset Parameters (page 1773) dialog box, modify station values, transition types, or other parameters.

**To grip edit an offset or widening**

1  Click the offset alignment or widening. Grips appear along the object.

2  Click the grip that adjusts the offset or widening as desired.

## Quick Reference

Ribbon

Click the offset alignment. Click Alignment tab ➤ Modify panel ➤ Offset Parameters 

Object Shortcut Menu

Right-click alignment or widening. Click Edit Offset Parameters

Command Line

EditOffsetAlignParameters

# Applying Design Criteria to an Existing Alignment

Use the Alignment Properties dialog box to apply design criteria to an existing alignment.

The criteria-based design feature provides the ability to verify that your alignment design meets the minimum standards required by your local agency.

For more information, see Criteria-Based Alignment Design (page 866).

**To apply design criteria to an existing alignment**

1   Click the alignment. Click Alignment tab ➤ Modify panel ➤ Alignment Properties
    drop-down ➤ Alignment Properties  .

2   In the Alignment Properties dialog box, click the Design Criteria (page 1754) tab.

    **NOTE** To use criteria-based design, an alignment must have at least one design speed.

3   In the Design Speeds section, click  Add Design Speed.

4   Specify the Start Station and Design Speed values.

5   Repeat Steps 3 and 4 to specify design speeds at other stations.

    An alignment can have unlimited numbers of design speeds, but can have only one speed at any given station. When a station has a design speed setting, that setting applies to either the remainder of the alignment or the next station at which a design speed is set. If a single entity has multiple speeds assigned to it, the highest speed is used to validate that entity against the design criteria. The lower speed values are ignored.

6   On the right side of the dialog box, select the Use Criteria-Based Design check box.

7   Specify the following settings:

    ■   Use Design Criteria File
        Select this check box to associate a design criteria file with the alignment. The default design criteria file location and the Default Criteria properties become available when you select the check box.

        **NOTE** The first design criteria file found in the *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>* directory is used by default when the criteria-based design feature is used.

    ■   Use Design Check Set
        Select this check box to associate a design check set with the alignment. The design check set list becomes available when you select the check box. Select a design check set from the list.

8   Click OK.

    **NOTE** The entire alignment design is validated when you click either Apply or OK.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Alignment Properties drop-down ➤ Alignment Properties

Toolspace Shortcut Menu

Prospector tab: Right-click <alignment item > ➤ Properties

Object Shortcut Menu

Right-click <alignment object> ➤ Alignment Properties

Command Line

EditAlignmentProperties

Dialog Box

Alignment Properties Dialog Box (page 1752)

# Viewing and Correcting Alignment Design Criteria Violations

Use the warning symbols to verify whether the alignment design meets the specified criteria and design checks.

When an alignment uses the criteria-based design feature, the alignment geometry must meet the design criteria specified in the design criteria file (page 2472) and design checks (page 2472).

When an alignment sub-entity violates the design criteria, a warning symbol is displayed in the following locations:

■ The drawing window — The ⚠ warning symbol is displayed on each sub-entity that violates the design criteria.

> **NOTE** The size of the warning symbol does not adjust automatically. After zooming in or out, enter REGEN to resize the warning symbols.

■ The Alignment Entities vista — The ⚠ warning symbol is displayed in the No. column of the sub-entity that violates the design criteria. If a criteria in the design criteria file was violated, the warning symbol is also displayed next to the specific criteria and parameter value that violates the design criteria.

■ The Alignment Layout Parameters dialog box — If a criteria in the design criteria file was violated, the ⚠ warning symbol is displayed in the Design Criteria panel next to the Property that was violated, and in the Layout Parameters panel next to the Parameter that was violated. If a design check was violated, the ⚠ warning symbol is displayed in the Design Checks panel next to the design check that was violated.

You can use the ⚠ warning symbols in the drawing window to view information about criteria and design checks that were violated. When the cursor is hovered over a warning symbol in the drawing window, a tooltip displays information about the violation. If a design criteria was violated, the tooltip displays the criteria that was violated, as well as the minimum value required to meet the criteria. If a design check was violated, the tooltip displays the name of the design check that was violated.

> **BEST PRACTICE** The name of the design check should be similar to the formula you enter in the Design Check field. When a design check is violated, only the design check name is displayed, and not the minimum acceptable value. If the formula is in the design check name, it is easier to correct the violation.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

## Turning Off the Alignment Warning Symbol

Change the visibility of the warning symbol that indicates violations in the alignment design.

There are several ways to turn off the warning symbol. The following methods are listed in the order in which they are recommended.

### Modify the Alignment Geometry

Edit the alignment design to meet the specified design criteria. This option clears the warning symbols from the drawing window, Alignment Entities vista, and Alignment Layout Parameters dialog box.

The criteria-based design feature does not validate portions of the alignment that have the design speed set to zero.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

For more information, see Correcting Alignment Design Criteria Violations (page 1010).

### Change the Alignment Style

Turn off the Warning Symbol display component in the current alignment style. Alternatively, create a separate alignment style that has the Warning Symbol display component turned off. This option clears the warning symbols from the drawing window for alignments that use that style. The warning symbols are still displayed in the Alignment Entities vista and Alignment Layout Parameters dialog box.

For more information, see Alignment Styles (page 885).

### Turn Off the AutoCAD Civil Solution Tip

The warning symbol is controlled by the Solution Tip setting in the Options dialog box on the AEC Editor tab. To hide the warning symbol for all drawings, clear the Drafting check box. This option clears all alignment and profile warning symbols from the drawing window. However, the warning symbols are still displayed in the Alignment Entities vista and Alignment Layout Parameters dialog box.

For more information, see AEC Editor Tab (Options Dialog Box) (page 2453).

**WARNING** Settings in the Options dialog box affect the overall behavior of AutoCAD Civil. If you clear the Solution Tip Drafting check box, the warning symbol for all alignment and profile objects will be hidden in every drawing that is opened on the current installation of AutoCAD Civil.

### Turn Off the Criteria-Based Design Feature

Disassociate the design criteria from the alignment. The alignment design check set and/or the design criteria file can be turned off in the Alignment Properties dialog box. This option clears the warning symbols from the drawing window, Alignment Entities vista, and Alignment Layout Parameters dialog box.

For more information, see Applying Design Criteria to an Existing Alignment (page 1007).

**WARNING** If you turn off the criteria-based design feature, the alignment design will not be validated against any design standards.

# Correcting Alignment Design Criteria Violations

Validate the alignment geometry against the specified criteria and design checks.

When an alignment uses the criteria-based design feature, the alignment geometry must meet the design criteria specified in the design criteria file (page 2472) and design checks (page 2472).

When an alignment sub-entity violates the design criteria, a warning symbol is displayed in several locations. For information about where the warning symbols appear, see Viewing and Correcting Alignment Design Criteria Violations (page 1008).

You can use the ⚠ warning symbols in the drawing window to view information about criteria and design checks that were violated. When the cursor is hovered over a warning symbol in the drawing window, a tooltip displays information about the violation.

- If a design criteria was violated, the tooltip displays the criteria that was violated, as well as the minimum value required to meet the criteria.

- If a design check was violated, the tooltip displays the name and formula of the design check that was violated.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

**To correct alignment design criteria violations using grips**

1  In the drawing window, hover the cursor over a ⚠ warning symbol.

   A tooltip displays information about the violation.

2  Select the alignment.

3  Use the alignment grips to modify the alignment geometry.

   The ⚠ warning symbol disappears when the sub-entity meets the design criteria.

   **NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed on the sub-entity. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria.

**To correct alignment design criteria violations by entering numeric values**

1  Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

2  On the Alignment Layout Tools toolbar, click ▦ Alignment Grid View.

3  In the Alignment Entities vista, click the row that contains the ⚠ warning symbol in the No. column.

   The sub-entity parameters are displayed in the Alignment Layout Parameters dialog box.

4  In the Alignment Layout Parameters dialog box, locate the ⚠ warning symbol in either of the following panels:

   - **Layout Parameters panel** — In the Value column, enter a numeric value that meets the value specified in the Constraints column.

■ **Design Checks panel** — Notice that the design check that has been violated. Modify the alignment design to meet the parameters specified in the design check.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏

Menu

Click Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar Icons

▦ Alignment Grid View

Dialog Box

# Reversing Alignment Direction

Use this command to reverse the direction and stationing of an alignment.

All objects that are dependent on the alignment (such as profiles) will also be updated.
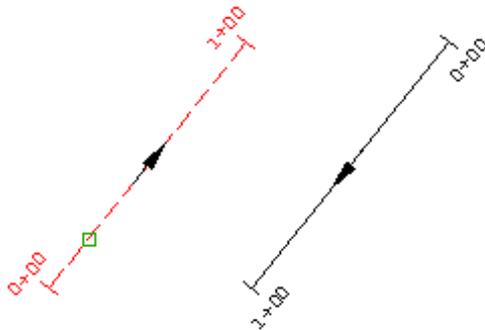


When you create an alignment, you can control the direction of the alignment, regardless of the direction the polyline was drawn. If you create an alignment using the Create Alignment From Polyline command, the alignment direction starts from the nearest endpoint you click the polyline. If you create an alignment using the Create Alignment By Layout command, the alignment direction is determined by the direction in which you draw the alignment. If you must change the direction of the alignment, you can use the Reverse Alignment Direction command. However, it is important to note that all objects that are dependent on the alignment (Profiles, Sample Lines, Sections, and Corridor objects, Superelevation) will change as well.

You can only change the direction of an alignment that is continuous.

You can also reverse the direction of an individual fixed, unattached alignment sub-entity. See Reversing Alignment Sub-Entity Direction (page 1012) for more information.

**To reverse the direction of an alignment**

➤ Click the alignment. Click Alignment tab ➤ Modify panel ➤ ▼ ➤ Reverse Direction ⇄.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ ▼ ➤ Reverse Direction ⇄

Menu

Alignments menu ➤ Reverse Alignment Direction

Command Line

ReverseAlignmentDirection

# Reversing Alignment Sub-Entity Direction

Reverse the direction of a fixed, unconnected line or curve entity.

This command works only on fixed entities that are not part of the solved alignment geometry. To reverse the direction of the entire alignment, use the ReverseAlignDirection command.
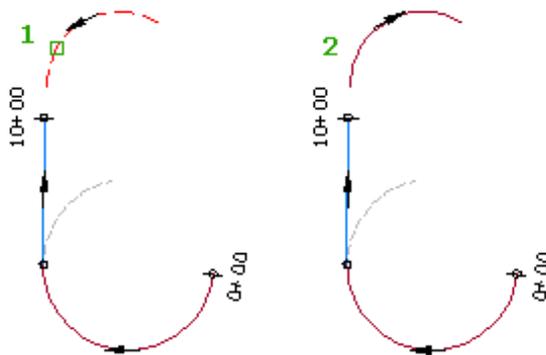


See Reversing Alignment Direction (page 1011) for more information.

When you add an alignment sub-entity to an existing alignment, the sub-entity direction is determined by the direction in which it was drawn. If the sub-entity direction is different from the alignment, the sub-entity cannot become part of the solved alignment geometry. The sub-entity cannot be included in the alignment stationing or labeling unless its direction matches the alignment's.

**To reverse the direction of an alignment sub-entity**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✎ .

**2** On the Alignment Layout Tools (page 1747) toolbar, click ⇙ Reverse Sub-Entity Direction.

**3** Click the alignment sub-entity in the drawing window.

### Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor ✏

Menu

Click Alignments  menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

⇙ Reverse Sub-Entity Direction

# Using AutoCAD Object Snaps with Alignment Objects

You can use AutoCAD object snaps (OSNAPs) with the alignment object. Although there are differences, alignment OSNAPs function in a manner consistent with OSNAPs to AutoCAD lines, curves, and polylines.

Node snaps apply to all pass-through points and all points of intersection.

**OSNAPs that apply to line segments**

Use OSNAPs with some alignment components. Alignment line components can have solved and unsolved segments, depending on whether they are fixed, free, or floating.(Yes indicates that OSNAPs do apply; No indicates that OSNAPs do not apply.)

| Line Segments | Solved Segments | Unsolved Segments |
| --- | --- | --- |
| Endpoint | Yes | No |
| Midpoint | Yes | No |
| Center | Yes | No |
| Node | Yes | No (Exception: Node does apply to visible pass-through points.) |
| Quadrant | Yes | No |
| Intersection | Yes | No |
| Extension | Yes | No |
| Insertion | Yes | No |
| Perpendicular | Yes | No |

| Line Segments | Solved Segments | Unsolved Segments |
|---|---|---|
| Tangent | Yes | No |
| Nearest | Yes | No |
| Apparent Intersection | Yes | No |
| Parallel | Yes | No |

**OSNAPs that apply to curve segments**

Use OSNAPS with some curve segments. Curves can have solved or unsolved segments depending on whether they are fixed, free, or floating curves.

(Yes indicates that OSNAPS apply; No indicates they do not apply.)

| Curve Segments | Solved Segments | Unsolved Segments |
|---|---|---|
| Endpoint | Yes | No |
| Midpoint | Yes | No |
| Center | Yes | No |
| Node | Yes | No (Exception: Node does apply to visible pass-through points.) |
| Quadrant | No | No |
| Intersection | Yes | No |
| Extension | Yes | No |
| Insertion | Yes | No |
| Perpendicular | Yes | No |
| Tangent | Yes | No |
| Nearest | Yes | No |
| Apparent Intersection | Yes | No |
| Parallel | Yes | No |

**OSNAPs that apply to spiral segments**

(Yes indicates that OSNAPS apply; No indicates they do not apply.)

| Spiral Segments | Solved Segment | Unsolved Segments |
|---|---|---|
| Endpoint | Yes | No |
| Midpoint | Yes | No |

| Spiral Segments | Solved Segment | Unsolved Segments |
| --- | --- | --- |
| Center | Yes | No |
| Node | Yes | No |
| Quadrant | No | No |
| Intersection | Yes | No |
| Extension | Yes | No |
| Insertion | Yes | No |
| Perpendicular | Yes | No |
| Tangent | Yes | No |
| Nearest | Yes | No |
| Apparent Intersection | Yes | No |
| Parallel | Yes | No |

# Deleting Alignment Entities

Use one of several methods to delete an alignment or alignment sub-entity.

The dependencies for deleting an alignment or alignment sub-entity are explained in the following topics.

# Deleting an Alignment

Delete an alignment from either the drawing window or the Toolspace Prospector tab.

If other objects were created from an alignment, you cannot use Prospector to delete the alignment. You can delete the alignment from the drawing window, but the dependant objects will also be deleted. Depending on the object type, the objects can also be deleted from Prospector.

For example, a profile and corridor can be created from an alignment. If you right-click the alignment in the Prospector, the Delete option is not available. If you select the alignment in the drawing window and press Delete, the corridor and profile are cleared from the drawing window. The profile is also deleted from Prospector because it is a child of the alignment object. The corridor object is still present in Prospector because the alignment was only one of the objects from which it was created. You can add another alignment to the corridor as a baseline, and then rebuild the corridor.

**To delete an alignment from the drawing window**

1   Select an alignment in the drawing.

2   Press Delete.

**To delete an alignment from the Prospector**

1   In Toolspace, on the Prospector tab, select the alignment object.

2   Right-click. Click Delete.

# Deleting an Alignment Sub-Entity

Delete a specified alignment entity.

You can delete an alignment sub-entity only if no other sub-entities use it as a defining parameter. The following points are the basic conditions for deleting the different types of alignment sub-entities:

- A fixed entity cannot be deleted if it has a floating or free entity attached to it.

- A floating entity cannot be deleted if it has either a floating or free entity attached to it.

- Free entities are not used to define the geometry of other entities. You can delete a free entity at any time.

**To delete an alignment sub-entity from the Alignment Layout Tools toolbar**

1 Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor .

2 On the Alignment Layout Tools toolbar, click Delete Sub-Entity.

3 Click the sub-entity in the drawing.

4 Press Enter to end the Delete Sub-Entity command.

**To delete an alignment sub-entity using the Delete key**

1 Ctrl+click an alignment sub-entity in the drawing.

**NOTE** For more information about selecting alignment sub-entities, see Selecting Alignment Entities (page 989).

2 Press Delete.

## Quick Reference

Ribbon

Click the alignment. Click Alignment tab ➤ Modify panel ➤ Geometry Editor

Menu

Click Alignments menu ➤ Edit Alignment Geometry

Command Line

EditAlignment

Alignment Layout Tools Toolbar icon

Delete Sub-Entity

# Alignment Command Reference

You can use AutoCAD Civil commands when you work with alignments. The following table lists the alignment-related AutoCAD Civil commands and briefly describes their functionality.

For more information about a command, follow the link in the Description column.

| Command | Description |
| --- | --- |
| AddAlignmentCurveTable | Adds an alignment curve table (page 987) |
| AddAlignLabels | Opens the Add Labels dialog box (page 971) |
| AddAlignmentLineTable | Adds an alignment line table (page 987) |
| AddAlignmentSegmentTable | Adds an alignment segment table (page 987) |
| AddAlignmentSpiralTable | Adds an alignment spiral table (page 987) |
| AddAlignOffLbl | Labels fixed station offsets (page 984) |
| AddAlignOffXYLbl | Labels multiple station offsets (page 983) |
| AddAlignSegLbl | Labels multiple segment labels (page 982) |
| AddAlignSegLbls | Labels multiple segment labels (page 982) |
| AddAlignTangentIntLbl | Labels single tangents intersections (page 984) |
| AddAlignTangentIntLbls | Labels multiple tangent intersections (page 985) |
| AddWidening | Adds a widened region to an existing alignment (page 896) |
| CreateAlignmentEntities | Opens the Create Alignment from Objects dialog box (page 1760) |
| CreateAlignmentLayout | Opens the Create Alignment-Layout dialog box (page 887) |
| CreateAlignmentReference | Creates an alignment reference in the current drawing (page 893) |
| CreateOffsetAlignment | Creates one or more offset alignments based on an existing alignment (page 895) |
| DesignCriteriaEditor | Opens the Design Criteria Editor dialog box (page 867) |
| EditAlignment | Opens the Alignment Layout Tools toolbar (page 988) |
| EditAlignmentLabels | Opens the Alignment Labels dialog box (page 973) |
| EditAlignmentTagNumbers | Renumbers alignment entity label tags (page 1544) |

| Command | Description |
| --- | --- |
| EditOffsetAlignParameters | Edits the parameters of an offset alignment or widening (page 1005) |
| ReverseAlignmentDirection | Reverses an alignment direction (page 1011) |

# Profiles

# 24

The main purpose of a profile is to show surface elevations along a horizontal alignment.

Use profiles to visualize the terrain along a route of interest or across a particular region.

For information on recommended workflows when working with profiles, see the *AutoCAD Civil Workflow Guide*.

## Understanding the Profile Object

There are several types of profiles: surface profiles, layout profiles, superimposed profiles, quick profiles, and corridor profiles.

- A surface profile—often called an existing ground (EG) profile—is extracted from a surface, showing the changes in elevation along a particular route.

- A layout profile, by contrast, is a designed object that shows the proposed elevation changes to be constructed. The layout profile—often called a design profile or finished grade (FG) profile—is typically used for a roadway or other graded site. For a road, the layout profile can include slopes and curves that are designed for safe driving at a particular speed.

- A superimposed profile is a profile of an alignment superimposed onto profile views of different alignments. These profiles are always dynamic and update when changes are made in the originating profile/alignment.

- A quick profile is a temporary object that shows the elevations along any existing line or series of selected points.

- A corridor profile is created from a corridor feature line, such as an edge of pavement. This profile is displayed on the profile view of the baseline alignment from which it is derived.

Layout profiles use two types of curves: crest curves and sag curves. Crest curves are placed on hilltops or wherever the grade changes to a lesser value. There are three types of crest curves: a positive to negative grade transition, positive to positive, and negative to negative:

**Sample Crest Curves**
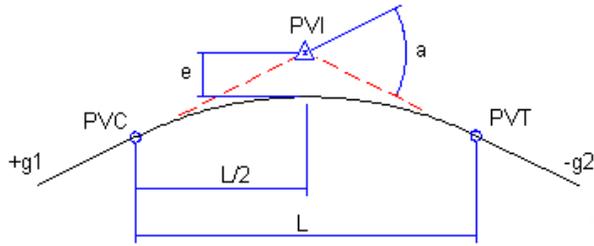


type 1

g1 and g2 - tangent grades in percent
a - algebraic difference
L - length of vertical curve
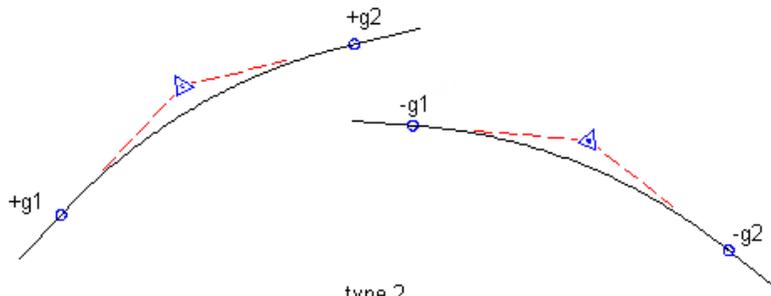


type 2

Sag curves are placed in valleys or wherever the grade changes to a greater value. There are three types of sag curves: a negative to positive grade transition, negative to negative, and positive to positive:

**Sample Sag Curves**



type 3
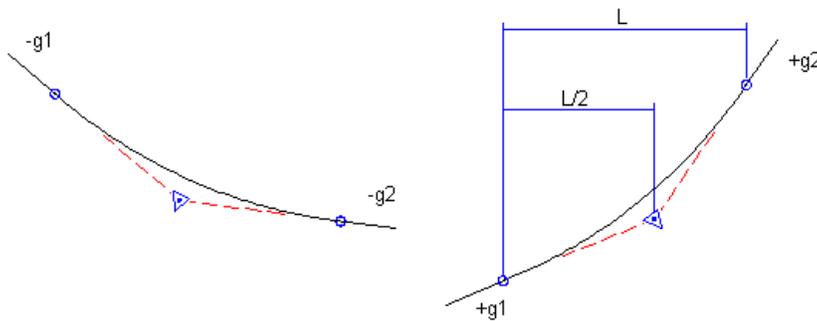


type 4

An offset profile is another type of profile commonly used in road design. While the road centerline provides the main horizontal alignment, various lines offset from the centerline mark other linear features, such as

edges of pavement, ditches, and sidewalks. Profiles along these offsets can be analyzed in relation to each other and to the centerline profile for a more complete view of the surface along a corridor. Offset profiles are created and managed independently from any offset alignments that may exist, though both can be used together in the design process.

When you create a surface profile, you specify whether it is dynamic or static. A dynamic profile automatically changes if the surface elevation changes. Such changes can occur if you move the horizontal alignment or edit the surface. A static profile represents the terrain at the time it is created, and does not respond to changes in the surface.

Profiles are displayed as graphed lines in a grid known as a profile view. Typically, you create a profile view to display surface profiles. Then, you draw layout profiles on the same grid to show elevation differences between the two surfaces. Profiles can be created and saved in a drawing, and displayed later when a profile view is created.

You can use a quick profile to view surface elevations along an object or through a selection of points. This quick profile is for temporary use and is not saved in the drawing. For more information, see Creating a Quick Profile (page 1058).

# The Profile View Object

Use a profile view to display profiles as graphed lines on a grid.

When you create a profile view, you specify which existing profiles to display on the grid. Use these profiles as a reference for drawing new layout profiles on the grid.

A profile view can include one or more related profiles, along with multiple data bands along the X-axis, above or below the grid. Data bands annotate the profiles with stationing, elevation, horizontal geometry, and other data that assists engineering analysis.

You typically use a profile view to display several profiles along a proposed route for a road, pipe, fence, or a similar structure. Use profile views to compare elevations of several surfaces or design profiles along the alignment.

Within a profile view, you can superimpose the profile of another alignment. For example, in the profile view for a road, you can superimpose the profile of a culvert that occupies the same corridor. By superimposing a profile, you can analyze culvert elevations in relation to the same alignment stations used for the road surfaces.

A profile view can include projected objects (page 2480) from the drawing that you want to see in relation to a profile. See Adding Projected Objects to a Profile View (page 1119) for more information.

If you design a road, pipeline, or a similar structure across the landscape, and you want to compare the feasibility of several possible routes, a profile view of each route can help with the comparison.

AutoCAD Civil allows profiles to split to fit within a specified profile view height and displayed in either single or multiple profile views. See Working with Profile Views (page 1106) for more information.

# Profile Object Relationships

A profile object is the child of a horizontal alignment. The horizontal alignment must exist to define the route of a profile across the terrain.

If you edit a horizontal alignment after creating dynamic surface profiles along its length, the profiles automatically change.

A profile view object is also dependent on a horizontal alignment. The length of the alignment can control the horizontal extents of the profile view grid, and the alignment stationing controls the annotation of horizontal axes. The vertical extents of the profile view have an optional relationship with one of the profiles in the profile view. You can set the vertical extents to a fixed value, but normally they are dynamically

linked to one of the profiles. The dynamic link ensures that the profile view always has several grid lines above and below the profiles.

A profile view can contain projected objects such as points, feature lines, or AutoCAD blocks. These objects exist in plan view and are projected into a profile view so that you can visualize them in relation to a profile.

# Profiles Collection (Prospector Tab)

Use the Profiles collection to access profiles in drawings.

As profiles are created, they are displayed as named objects in the Toolspace Prospector tab under the Profiles collection.

Right-click the Profiles collection to do any of the following:

- Export site data to LandXML. (page 1682)
- Recursively refresh the Profiles collection and list view.

If one or more profiles exist in the current drawing, expand the Profiles collection in Toolspace on the Prospector tab. The Profiles collection displays the names of all profiles associated with a particular horizontal alignment, as well as a tabular list of the profiles.

Right-click an individual profile name to do any of the following:

- Review or change properties (page 1024) of the profile.
- Export profile data to LandXML (page 1682).
- Delete the profile from the drawing.
- Refresh the display of the profile in the Prospector tab.

# Profile Views Collection (Prospector Tab)

Use the Profile Views collection to access profile views in drawings.

As profile views are created, they are displayed as objects under the Profile Views collection.

Right-click the Profile Views collection to refresh the Profile Views collection and list view.

If one or more profile views exist in the current drawing, expand the Profile Views collection in Toolspace on the Prospector tab. The Profile Views collection displays the names of all profile views associated with a particular horizontal alignment, as well as a tabular list of the profile views.

Right-click an individual profile view name to:

- Review or change properties (page 1024) of the profile view.
- Delete the profile view.
- Refresh the display of the profile view in the Prospector tab.

# Profile Collection (Settings Tab)

Use the Profile collection on the Settings tree to manage profile settings, profile styles, profile label styles, and profile command settings.

Right-click the Profile collection to do any of the following:

- Edit the profile feature settings.

- Edit the profile label style defaults.

- Refresh the display of the Profile collection in the Prospector tab.

Expand the Profile collection to display and edit the styles and command settings that are available for profiles.

| For more information about... | See... |
| --- | --- |
| Profile Settings | Profile Settings (page 1025) |
| Profile Styles | Profile Styles (page 1027) |
| Profile Label Styles | Managing Profile Label Styles (page 1034) |
| Profile Commands | Profile Command Reference (page 1121) |

## Profile View Collection (Settings Tab)

Use the Profile View collection on the Settings tree to manage settings, styles, label styles, band styles, and command settings for profile views.

Right-click the Profile View collection to do any of the following:

- Edit the profile view feature settings

- Edit the profile view label style defaults

- Refresh the display of the Profile Views collection in the Prospector tab

Expand the Profile Views collection to display and edit the styles and command settings that are available for profile views.

| For more information about... | See... |
| --- | --- |
| Profile View Styles | Profile View Styles (page 1029) |
| Profile View Label Styles | Profile Labels and Label Styles (page 1033) |
| Data Band Styles | Profile View Band Styles (page 1038) |
| Profile View Commands | Profile View Command Reference (page 1121) |

## Profile Shortcut Menu

Use the profile shortcut menu for quick access to common functions.

When you right-click a profile in the drawing, the shortcut menu is displayed.

The following table explains the items on the menu that are specific to profiles. Other selections on the menu are standard AutoCAD options.

| Use this menu item... | To... |
| --- | --- |
| Profile Properties | Open the Profile Properties Dialog Box (page 2174) for the selected profile. |
| Edit Profile Style | Open the Profile Style Dialog Box (page 2176). |

| Use this menu item... | To... |
| --- | --- |
| Edit Profile Geometry | Open the Profile Layout Tools (page 2170). |
| Superimposed Options | Open the Superimpose Profile Options Dialog Box (page 2193). Available for superimposed profiles only. |
| Edit Labels | Open the Profile Label Set Dialog Box (page 2167) |
| Inquiry | Open the Inquiry Tool Dialog Box (page 2462) with a Profile inquiry selected |

## Profile View Shortcut Menu

Use the profile view shortcut menu for quick access to common functions.

When you right-click a profile view grid in the drawing, the shortcut menu is displayed.

The following table explains the items on the menu that are specific to profile views. Other selections on the menu are standard AutoCAD options.

| Use this menu item... | To... |
| --- | --- |
| Profile View Properties | Open the Profile View Properties Dialog Box (page 2178). |
| Edit Profile View Style | Open the Profile View Style Dialog Box (page 2185). |
| Inquiry | Open the Inquiry Tool Dialog Box (page 2462) with a Profile inquiry selected. |

## Projected Object Shortcut Menu

Use the shortcut menu for a projected object for quick access to object properties.

When you right-click a projected object (page 2480)in a profile view, the shortcut menu is displayed. The only menu selection specific to the projection is Projection Object Properties. It opens the Projections tab (page 2183) of the Profile View Properties dialog box.

## Profile Properties

When you create a profile, you set the properties, including its name, style, station range, and labeling.

Use the Profile Properties Dialog Box (page 2174) to review or edit the name and other information for an existing profile. The three tabs correspond to general types of properties: information, profile data, and design criteria.

**See also:**

■ Setting Profile Properties (page 1041)

## Profile View Properties

By specifying properties, you control the content and format of a profile view.

Set these properties when you create a profile view. At any time, you can use the Profile View Properties Dialog Box (page 2178) to make changes. The tabs on this dialog box correspond to general types of properties: Information, Stations, Elevations, Profiles, Bands, and Hatch. Additional tabs for Projections and Pipe Networks appear if the profile view includes these objects.

**See also:**

- Setting Profile View Properties (page 1042)

# Settings for Profiles and Profile Views

You can review and edit the default style and name format settings for profiles and profile views. You can also review and edit default settings for profile and profile view commands.

Settings are handled in a standard way throughout AutoCAD Civil through the Settings tree at three levels: drawing, feature, and command. For more information, see Understanding Settings (page 61).

# Profile Settings

You can review and edit the default style and name format settings for profiles and profile commands.

Settings are handled in a standard way throughout AutoCAD Civil, and accessed in Toolspace on the Settings tab. Control settings at three levels: the drawing, the feature, and the command. For more information, see Understanding Settings (page 61).

On the Settings tab, use the Profile collection shortcut menu to establish default settings for all profiles. At this level, you can either change profile-specific settings or override the drawing ambient settings.

Use the Profile Commands collection under the Profile collection to change settings for a specific command. At this level, you can either change settings for profile commands or override the settings at the profile feature level.

**NOTE** Overrides to the drawing ambient settings at the Profile collection level and the Profile Command collection level affect only the specified level. The drawing level settings are not changed. For more information, see The Commands Collections (Settings Tree) (page 97).

This section describes only the default styles and name format settings for profile commands. It does not cover the drawing ambient settings, even though these settings are displayed in the Edit Feature Settings dialog box. For more information about the drawing ambient settings, see Specifying Drawing Settings (page 63).

For more information about profile view settings, see Profile View Settings (page 1026).

**To change profile settings**

1 In Toolspace, on the Settings tab, do one of the following:
   - Right-click the Profile collection and click Edit Feature Settings
   - In the Profile collection, open the Commands folder, right-click the command for which you want to change settings. Click Edit Command Settings.

   In the Edit Feature (or Command) Settings dialog box, profile feature-level settings are marked with ⌄ . Profile command-level settings are marked with ⊞✓ .

2 Expand the default styles and default name format to see the current settings.

**3** In the Value column, click the entry you want to change. A small Browse button appears on the right side of the cell.

**4** Click the Browse button to open a dialog box where you can modify the value.

**5** In the Edit Feature (or Command) Settings dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Profile collection ➤ Edit Feature Settings
Settings tab: Profile ➤ Commands ➤ right-click command name ➤ Edit Command Settings

# Profile View Settings

You can review and edit the default style and name format settings for profile views and profile view commands.

Settings are handled in a standard way throughout AutoCAD Civil, and accessed in Toolspace on the Settings tab. You can control settings at three levels: the drawing, the feature, and the command. For more information, see Understanding Settings (page 61).

On the Settings tab, use the Profile View collection shortcut menu to establish default settings for all profile views. At this level, you can change settings specific to profile views, which will override the drawing ambient settings for profile view.

Use the Commands collection under the Profile View collection to change settings for a specific command. At this level, you can change settings specific to profile view commands or override the settings at the level of the profile view feature.

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**NOTE** Overrides to the drawing ambient settings at the Profile View collection level and the Profile View Command collection level affect only the specified level. The drawing level settings are not changed. For more information, see The Commands Collections (Settings Tree) (page 97).

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This section describes only the default styles and name format settings for profile view commands. It does not cover the drawing ambient settings, even though these settings are displayed in the Edit Feature Settings dialog box. For more information about the drawing ambient settings, see Specifying Drawing Settings (page 63).

For more information about profile settings, see Profile Settings (page 1025).

**To change profile view settings**

**1** In Toolspace, on the Settings tab, do one of the following:

■ Right-click the Profile View collection and click Edit Feature Settings

■ In the Profile View collection, open the Commands folder, right-click the command for which you want to change settings. Click Edit Command Settings.

In the Edit Feature (or Command) Settings dialog box, profile view feature-level settings are marked

with ⌄ . Profile view command-level settings are marked with ▦✔ .

**2** Expand the default styles and default name format to see the current settings.

**3** In the Value column, click the entry you want to change. A small Browse button is displayed on the right side of the cell.

**4** Click the Browse button to open a dialog box where you can modify the value.

**5** In the Edit Feature (or Command) Settings dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Profile View collection ➤ Edit Feature Settings
Settings tab: Profile View ➤ Commands ➤ right-click <command name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Profile View Dialog Box (page 2159)

# Styles and Display of Profiles and Profile Views

Styles control the way a profile and profile view are displayed in a drawing. Profiles have label styles that control the appearance and behavior of their labels in a drawing.

# Profile Styles

You can use styles to control profile appearance in model, profile, and section views.

Use the Toolspace Settings tree to manage profile styles. All objects have an object style collection on the Settings tab that you can use to create, edit, copy, and delete the styles for that object. For more information, see The Object Style Collection (Settings Tree) (page 94).

To examine the properties of an existing style, follow the procedure for Creating and Editing Profile Styles (page 1027).

A profile object can reference two style types:

- **Profile Style.** This style controls the visibility of profile components and their rendering in both 2 and 3 dimensions.

- **Profile Label Style**. This style controls the appearance and content of labels for stations, lines, curves, and geometry points along the profile. For more information, see Profile Labels and Label Styles (page 1033).

## Creating and Editing Profile Styles

Use the Toolspace Settings tab to create or edit a profile style.

Create a style by adjusting the default settings, or by copying an existing style and changing its properties to suit your requirements.

You can choose markers for key geometry points on layout profiles.

Set the style properties on the following tabs in the Profile Style dialog box:

- **Information.** Specify the style name and description. See also Information Tab (Style Dialog Box) (page 1801).

- **Design.** Specify the tessellation distance to control the accuracy of rendering profile curves in 3D view.

- **Markers.** Specify the markers for locations on layout profiles such as PVI points, pass-through points, and profile start and end.

- **Display.** Specify the display attributes of profile components, including visibility, layer, color, linetype, line weight, linetype scale, and plot style. See also Display Tab (Style Dialog Box) (page 1801).

- **Summary**. Provides a concise display of the style properties. See also Summary Tab (Style Dialog Box) (page 1803).

**TIP** Any style changes you make apply to all existing profiles in the drawing that use the style. To change the style of some profiles but not all, consider creating a separate style.

**See also:**

- Profile Style Dialog Box (page 2176)

**To create a profile style**

1 In Toolspace, on the Settings tab, expand the Profile Styles collection. Right-click and click New.

2 In the Profile Style dialog box (page 2176), specify the name of the style and other settings as required.

3 Click OK.

**To copy a profile style**

1 In Toolspace, on the Settings tab, expand the Profile Styles collection. Identify the style you want to copy as a basis for the new style.

2 Right-click the style name and click Copy.

3 In the Profile Style dialog box (page 2176), change the name of the style and other settings as required.

4 Click OK.

**To edit a profile style**

1 In Toolspace, on the Settings tab, expand the Profile Styles collection. Identify the style you want to edit.

2 Right-click the style name and click Edit.

3 In the Profile Style dialog box (page 2176), change settings as required.

4 Click OK.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Edit Profile Style ⌄◻

Toolspace Shortcut Menu

Create: Settings tab, right-click Profile Styles collection ➤ New
Copy: Settings tab, Profile ➤ Profile Styles ➤ right-click <style name> ➤ Copy
Edit: Settings tab, Profile ➤ Profile Styles ➤ right-click <style name> ➤ Edit

Command Line

CreateProfileStyle

EditProfileStyle

Dialog Box

Profile Style (page 2176)

## Selecting a Different Profile Style

Change the appearance of a profile by changing its style.

The profile style controls the visibility of profile components and their rendering in both 2 and 3 dimensions.

**To select a different profile style**

1  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Profile Properties ![icon] .

2  In the Profile Properties dialog box, on the Information tab (page 2174), select a style from the Object Style list.

3  Click OK.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Profile Properties ![icon]

Toolspace Prospector Tab Shortcut Menu

Alignments ➤ Profiles collection ➤ right-click <profile name> ➤ Properties

Drawing Window Shortcut Menu

Right-click profile object ➤ Profile Properties

Command Line

EditProfileProperties

Dialog Box

Profile Properties Dialog Box (page 2174)

# Profile View Styles

You can use profile view styles to control the appearance of the graph that displays a profile.

Use the Toolspace Settings tab to manage profile view styles. All objects have an object style collection on the Settings tab that you use to create, edit, copy, and delete the styles for that object. For more information, see The Object Style Collection (Settings Tree) (page 94).

To examine the properties of an existing style, follow the procedure for creating and editing profile view styles (page 1030).

A profile view object can reference the following style types:

- **Profile View Style.** This style controls the format of the graph on which the profiles are displayed, as well as the title and annotations on the axes. For more information, see Creating and Editing Profile View Styles (page 1030).

- **Label Style.** This style controls the format of two types of manually placed labels along the profile: station elevation and depth (elevation difference and/or grade between any two points). For more information, see Profile Labels and Label Styles (page 1033).

- **Band Style.** This style controls the format of data bands above and below the profile view. Data bands show station elevations, horizontal geometry, vertical geometry, superelevation, sectional data, and pipe data. For more information, see Profile View Band Styles (page 1038).
  For convenience, you can define a group of band styles as a band style set (page 1040). Then, apply the entire set of band styles to a profile view with a single action, rather than applying each band style separately.

- **Projection Style.** This multipurpose style controls the appearance of objects projected into profile views or section views.

- **Shape Style.** This multipurpose style controls the appearance of hatching for cut and fill areas between surfaces and profiles.

## Creating and Editing Profile View Styles

Use the Toolspace Settings tab to create or edit a profile view style.

You can choose which elements to display, and change the appearance of the graph, grid, title, axes ticks, and text.

The main properties are set on the following tabs in the Profile View Style dialog box:

- **Information.** Specify style name and description. See also Information Tab (Style Dialog Box) (page 1801).

- **Graph.** Specify profile view direction and vertical scale factor

- **Grid.** Specify grid clipping, padding, and offset options

- **Title Annotation.** Specify content and appearance of titles for the overall profile view and individual axes

- **Horizontal Axes.** Specify ticks, tick justification, tick and grid spacing, and annotation of the horizontal axes

- **Vertical Axes.** Specify ticks, tick justification, tick and grid spacing, and annotation of the vertical axes

- **Display.** Specify display attributes for profile view components, including visibility, layer, color, linetype, line weight, linetype scale, and plot style. See also Display Tab (Style Dialog Box) (page 1801).

- **Summary.** Provides a concise display of the style properties. See also Summary Tab (Style Dialog Box) (page 1803).

Before editing a style to change its properties, remember that any changes you make apply to all profile views that use the style. To change the style of some profile views but not all, consider creating a separate style.

**See also:**

- Profile View Style dialog box (page 2185)

**To create a profile view style**

1   In Toolspace, on the Settings tab, expand the Profile View Styles collection. Right-click and click New.

2   In the Profile View Style dialog box (page 2185), specify the name of the style and other settings as required.

3   Click OK.

**To copy a profile view style**

1   In Toolspace, on the Settings tab, expand the Profile View Styles collection. Identify the style you want to copy as a basis for the new style.

2   Right-click the style name and click Copy.

3   In the Profile View Style dialog box (page 2185), change the name of the style and other settings as required.

4   Click OK.

**To edit a profile view style**

1   In Toolspace, on the Settings tab, expand the Profile View Styles collection. Identify the style you want to edit.

2   Right-click the style name and click Edit.

3   In the Profile View Style dialog box (page 2185), change settings as required.

4   Click OK.

## Quick Reference

Toolspace Shortcut Menu

Create: Settings tab, right-click Profile View Styles ➤ New
Copy: Settings tab, Profile View ➤ Profile View Styles ➤ right-click <style name> ➤ Copy
Edit: Settings tab, Profile View ➤ Profile View Styles ➤ right-click <style name> ➤ Edit

Command Line

CreateProfileViewStyle
EditProfileViewStyle

Dialog Box

Profile View Style (page 2185)

## Creating and Editing Projection Styles

Use these general styles to control the display of AutoCAD objects projected into profile views and section views.

Within a style, the settings for the appearance of an object type in profile views are independent from the settings for section views.

The appearance of AutoCAD Civil projected objects is controlled by styles for the particular object type: points, feature lines, and survey figures.

**See also:**

- Display Tab (Point Style Dialog Box) (page 2117)
- Display Tab (Feature Line Style Dialog Box) (page 1898)
- Display Tab (Survey Figure Style Dialog Box) (page 2393)

**To create a projection style**

1 In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Right-click Projection Styles and click New.

2 In the Projection Styles dialog box (page 1807), specify the name of the style and other settings as required.

3 Click OK.

**To copy a projection style**

1 In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Identify the style you want to copy as a basis for the new style.

2 Right-click the style name and click Copy.

3 In the Projection Styles dialog box (page 1807), change the name of the style and other settings as required.

4 Click OK.

**To edit a projection style**

1 In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Identify the style you want to edit.

2 Right-click the style name and click Edit.

3 In the Projection Styles dialog box (page 1807), change settings as required.

4 Click OK.

## Quick Reference

Ribbon

Click a projected object. Click Projected Object tab ➤ Modify Projected Object panel ➤ Projection Properties drop-down ➤ Edit Style

Toolspace Shortcut Menu

Create: Settings tab, expand General ➤ Multipurpose Styles ➤ right-click Projection Styles ➤ New
Copy: Settings tab, expand General ➤ Multipurpose Styles ➤ Projection Styles ➤ right-click <style name> ➤ Copy
Edit: Settings tab, expand General ➤ Multipurpose Styles ➤ Projection Styles ➤ right-click <style name> ➤ Edit

Command Line

EditProjectionStyle

# Profile Labels and Label Styles

You can label profiles and profile views using different label types.

**Profile labels**

When you create a layout profile, you can automatically label it using label styles you specify in the Create Profile - Draw New dialog box (page 2142). For other types of profiles, add labels using the Edit Labels command.

Profile label styles can be configured to mark any of the following standard points along the profile:

- Major and minor stations of the parent horizontal alignment

- Horizontal geometry points

- Grade breaks

- Lines

- Sag curves

- Crest curves

You can save profile label styles as a profile label set and apply the labels to a profile as a group.

**See also:**

- Staggering Profile and Section Labels (page 1527)

**Profile view labels**

After creating a profile view, use the Add Labels dialog box (page 1970) to manually add labels to particular points of interest anywhere on the profile view grid. These labels can be of three types:

- Station Elevation, showing the station and elevation of a point.

- Depth, showing the elevation difference between two points, and other data such as slope and distance.

- Projection, showing the station and elevation of a projected object (page 2480).

**Profile view bands**

Place data bands along the top or bottom of a profile view grid to annotate the various profile lines with station and elevation data, vertical or horizontal geometry points, or superelevation, sectional, or pipe network data. You can also group a set of commonly used data bands into a band set that can be applied to a profile view as a single selection.

The topics in this section describe unique properties of profile labels. For a general overview of labels, see Understanding Labels (page 1466).

## Editing Profile Labels

Edit the set of labels used on a profile line.

Edit the settings for each label type, and import or save label sets.

Editing the labels on a profile does not affect instances of the same profile in other profile views.

Profile labels that display curve and tangent dimensions have editing grips that you can use to move labels.

Use the Profile Labels dialog box (page 2167) to numerically edit the position of labels, change the style of a label, or save your changes as a label style.

Label styles ensure convenience and consistency. A particular profile may require different labels. After the labels are edited, save the modified label set for reuse with other profiles.

**See also:**

■ Staggering Profile and Section Labels (page 1527)

**To edit profile labels**

1 Do one of the following:

■ Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Add/Edit Profile Labels ✎ . Select a profile in the drawing to open the Profile Labels dialog box.

■ Click the profile line you want to edit. Right-click and click Edit Labels

2 In the Profile Labels dialog box (page 2167), review the existing labels and their attributes. Do any of the following:

■ Add or remove labels of any type.

■ Change the style for a label type.

■ Import a standard set of labels.

■ Save your modified labels as a set to use again.

■ Stagger labels to avoid label collisions. For more information, see Staggering Profile and Section Labels (page 1527).

3 Click Apply to see your changes on the profile line.

4 Click OK to save the changes.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Add/Edit Profile Labels ✎

Menu

Click Profiles menu ➤ Add Profile View Labels ➤ Add/Edit Profile Labels.

Object Shortcut Menu

Click profile, right-click ➤ Edit Labels

Command Line

EditProfileLabels

Dialog Box

Profile Labels (page 2167)

## Managing Profile Label Styles

Use the Toolspace Settings tab to create and edit profile label styles.

The types of profile label styles that you can create:

| This type of profile label style... | Labels... |
| --- | --- |
| Major Stations | Stations along the profile at major intervals |
| Minor Stations | Stations along the profile at minor intervals |
| Horizontal Geometry Points | Locations where the horizontal alignment geometry changes, such as the start of a curve |
| Lines | Lines that form the tangents of a profile |
| Grade Breaks | Locations where the vertical grade changes |
| Sag Curves | Sag curves in the profile |
| Crest Curves | Crest curves in a profile |

When you need a new profile label style, it is usually easiest to find an existing style that is similar to what you need, then copy it and make the required changes. You can also create a style and change whatever default values are not suitable.

You can group several profile label styles into a label set. Apply the label set to a profile as a single setting. An existing label set can be edited or copied to create a new one.

Label spacing is an important design consideration for surface profiles. Wherever they cross uneven surfaces, this type of profile can have many short tangents and closely spaced grade breaks. As a result, labels for these features can overlap and become unreadable when they require more horizontal space than is available. To address this condition for one label type, the Profile Labels dialog box (page 2167) provides a weeding setting that suppresses grade break labels within a specified distance of each other.

Profile label styles are created and edited using the Label Style Composer. For more information, see Overview of the Label Style Composer (page 1473).

**To copy a profile label style**

1   In Toolspace, on the Settings tab, expand the Profile, Label Styles, and specific label style collection.

2   Right-click the name of the label style you want to copy, and click Copy. The Label Style Composer dialog box (page 1942) is displayed.

3   Change settings as required to produce the style you want. Specify a name for the style.

4   Click OK to save the style.

**To edit a profile label style**

1   In Toolspace, on the Settings tab, expand the Profile and Label Styles collections. Click a specific label style collection.

2   Right-click the name of the label style you want to edit and click Edit. The Label Style Composer dialog box (page 1942) is displayed.

3   Change settings as required.

4   Click OK to save the style.

**To create a profile label style**

1  In Toolspace, on the Settings tab, expand the Profile and Label Styles collections.

2  Right-click the collection of label style you want to create and click New. The Label Style Composer dialog box (page 1942) is displayed.

3  Change settings as required. Specify a name for the new style.

4  Click OK to save the style.

**To create a profile label set**

1  In Toolspace, on the Settings tab, expand the Profile, Label Styles, Label Sets collections.

2  Do one of the following:
   ■ Right-click the Label Sets collection and click New.
   ■ To copy an existing set as a starting point, right-click the set and click Copy

3  In the Label Set dialog box (page 1771), on the Information tab, specify a name for the label set. Optionally, specify a description.

4  On the Labels tab (page 1772), specify the individual labels to include in the set, and settings for each one.

5  Click OK to save the new set.

## Managing Profile View Label Styles

Use the Settings tab on Toolspace to create and edit profile view label styles.

Profile view labels are added manually to mark points of interest on a profile view. The following types of profile view labels are available:

■ **Station Elevation** labels show the elevation and station value at a specified point anywhere in the profile view.

■ **Depth** labels show the elevation difference between two specified points, and other selected data, such as slope and distance.

■ **Projection** labels show data about a projected object (page 2480), such as the elevation and station value.

Create and edit the styles for profile view labels using the Label Style Composer.

If a profile view contains projected objects, label styles for these objects are managed as feature settings for the profile view object.

**See also:**

■ Overview of the Label Style Composer (page 1473)

■ Edit Feature Settings - Profile View Dialog Box (page 2159)

**To copy a profile view label style**

1  In Toolspace, on the Settings tab, expand the Profile View, Label Styles, and specific label style collections.

2  Right-click the name of the label style you want to copy. Click Copy. The Label Style Composer dialog box (page 1942) is displayed.

**3** Change settings as required. Specify a name for the style.

**4** Click OK to save the style.

**To edit a profile view label style**

**1** In Toolspace, on the Settings tab, expand the Profile View, Label Styles, and specific label style collections.

**2** Right-click the name of the label style you want to edit. Click Edit. The Label Style Composer dialog box (page 1942) is displayed.

**3** Change settings as required.

**4** Click OK to save the style.

**To create a profile view label style**

**1** In Toolspace, on the Settings tab, expand the Profile View, Label Styles, and specific label style collections.

**2** Right-click the name of the label style you want to create, and click New. The Label Style Composer dialog box (page 1942) is displayed.

**3** Change settings as required. Specify a name for the new style.

**4** Click OK to save the style.

## Adding Profile View Labels

Manually place profile view labels at points of interest anywhere on a profile view.

You can label any point to show station and elevation, to show the relationship between two points, or to mark the location of a projected object (page 2480).

Station Elevation labels show the elevation and station value at any specified point in the profile view.

Depth labels show the elevation difference between two specified points, and other selected data, such as slope and distance.

Projection labels show information about an object projected into the profile view, such as its elevation and station.

Projection labels cannot be used with projected feature lines or 3D polylines that appear as lines in a profile view.

---

**TIP** To label points on a feature line projected into a profile view, use Station Elevation or Depth labels.

---

**To add a profile view station elevation label**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Station Elevation .

**2** Select the station and elevation to label.

**To add a profile view depth label**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Depth .

**2** Click any two locations. Labels are displayed, showing the depths, grade, or other data about the relationship between the two points.

**To add a projection label**

➤ Select the projected object. Right-click and click Add Label.

## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Add Profile View Labels

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Station Elevation

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Depth

Click Projected Object tab ➤ Labels panel ➤ Add Object Labels

Menu

Click Profiles  menu ➤ Add Profile View Labels ➤ Add Profile View Labels
Click Profiles menu ➤ Add Profile View Labels ➤ Station Elevation
Click Profiles  menu ➤ Add Profile View Labels ➤ Depth

Command Line

AddProfileViewLabels
AddProfileViewDepthLbl
AddProfileViewStaElevLbl
AddProjectionLabel

# Profile View Band Styles

Use the Toolspace Settings tab to create and edit profile view band styles.

Profile view band styles control the location and content of data bands.

Use the Profile View Properties dialog box (page 2178) to specify which data band styles to use and where to place them on the profile view.

Data band labels are label objects that can be manipulated like other AutoCAD Civil label objects. For more information, see Editing Data Band Labels (page 1529).

The types of data band styles that you can create:

| This type of data band style... | Provides this type of annotation... |
|---|---|
| Profile Data | Stations, elevation of one or two profiles at each station, and elevation difference between two profiles at the station |
| | **NOTE** The individual geometry points to label can be specified with the Profile Data band type. For more information, see Adding Data Bands to a Profile View (page 1115). |
| Vertical Geometry | Geometric details of vertical tangents and curves |
| Horizontal Geometry | Geometric details of horizontal tangents, curves, and spirals |

| This type of data band style... | Provides this type of annotation... |
| --- | --- |
| Superelevation | Critical transition stations and transition diagrams for the superelevation at curves |
| Sectional Data | Sampled station, material, typical section details, incremental distance and volume, balanced volume information |
| Pipe Data | Location and key dimensions of pipe network parts, offset, elevation, direction |

When you need a new data band style, it is usually easiest to find an existing style that is similar to what you need, then copy it and make the required changes. You can also create one and change whatever default values are not suitable.

The notes in this section describe what is unique to profile data bands. For a general overview of labels, see Understanding Labels (page 1466).

**See also:**

- Adding Data Bands to a Profile View (page 1115)
- Staggering Data Band Labels (page 1528)

**To copy a data band style**

1  In Toolspace, on the Settings tab, expand the Profile View, Band Styles, and specific band style collections.

2  Right-click the name of the band style you want to copy. Click Copy. The Profile Data Band Style Dialog Box (page 2164) is displayed.

3  On the Information tab (page 2164), enter a new name for the style. Optionally, add a description.

4  On the Band Detail tab (page 2165), compose the band title and modify any settings for the band title and layout.

5  On the right side of the Band Details tab, select a label location. Click Compose Label to define the property fields annotated by the band. For more information, see Layout Tab (Label Style Composer Dialog Box) (page 1944).

6  On the Display tab (page 2166), modify any display settings for the band components.

7  On the Summary tab, review all settings for the style. Make any corrections.

8  Click OK to save the style.

**To edit a data band style**

1  In Toolspace, on the Settings tab, expand the Profile View, Band Styles, and specific band style collections.

2  Right-click the name of the band style you want to change. Click Edit. The Profile Data Band Style dialog box (page 2164) is displayed.

3  Review the settings on the Information tab (page 2164), the Band Details tab (page 2165), and the Display tab (page 2166). Make any changes.

4  On the Summary tab, review all settings for the style. Make any corrections.

5  Click OK to save the style.

**To create a data band style**

**1** In Toolspace, on the Settings tab, expand Profile View and Band Styles.

**2** Right-click the name of the band type for which you want to create a style. Click New. The Profile Data Band Style dialog box (page 2164) is displayed.

**3** On the Information tab (page 2164), enter a new name for the style. Optionally, add a description.

**4** On the Band Details tab (page 2165), compose the band title and modify any settings for the band title and layout.

**5** On the right side of the Band Details tab, select a label location and click Compose Label to define the property fields annotated by the band. For more information, see Layout Tab (Label Style Composer Dialog Box) (page 1944).

**6** On the Display tab (page 2166), modify any display settings for the band components.

**7** On the Summary tab, review all settings for the style. Make any corrections.

**8** Click OK to save the style.

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: Profile View ➤ Band Styles

Dialog Box

  Profile Data Band Style (page 2164)

# Profile View Band Sets

Use a data band set to manage a group of band styles that you want to apply to other profile views.

**See also:**

■  Adding Data Bands to a Profile View (page 1115)

**To create a band set**

**1** In Toolspace, on the Settings tab, expand Profile View and Band Styles.

**2** Right-click the Band Sets folder, and click New. The Band Set dialog box (page 2139) is displayed.

**3** On the Information tab (page 2139), enter a name and optional description for the band style set.

**4** On the Bands tab (page 2140), in the List Of Bands area, specify the Location where you want to place the bands.

**5** In the Band Type field, select the type of band to add to the set.

**6** In the Select Band Style field, select an existing band style or use one of the style buttons to create a style.

**7** Click Add to add the band type to the List Of Bands.

**8** Repeat steps 4 to 7 to add more bands to the set.

**9** Use the three band position buttons on the right side of the List of Bands to put the data bands in the desired order and to delete unwanted bands.

**10** Click OK to save the band style set.

**To edit a band set**

**1** In Toolspace, on the Settings tab, expand the Profile View, Band Styles, and Band Sets collections.

**2** Right-click the name of the set you want to edit. Click Edit. The Band Set dialog box (page 2139) is displayed.

**3** Change content as required. Click OK to save the modified set.

**To copy a band set**

**1** In Toolspace, on the Settings tab, expand the Profile View, Band Styles, and Band Sets collections.

**2** Right-click the name of the set you want to copy. Click Copy. The Band Set dialog box (page 2139) is displayed.

**3** On the Information tab (page 2139), enter a new name. Optionally, enter a description.

**4** Change other content as required. Click OK to save the new set.

# Properties of Profiles and Profile Views

Several aspects of profile data and display are controlled by properties.

# Setting Profile Properties

Set properties of individual profiles either from the Toolspace Prospector tab, or from a profile view.

You can edit some of the properties, including those for design criteria.

Profile properties are organized on three tabs of the Profile Properties (page 2174) dialog box: information, profile data, and design criteria.

### Information

The profile name, description, and object style. By changing the style here you can instantly change the displayed appearance for the profile.

### Profile Data

Each profile is one of the following types: surface, layout, superimposed, or corridor.

A surface profile can be either static or dynamic. The elevation data in a dynamic profile changes to respond to changes in the parent alignment or surface. A static profile displays the elevation data at a particular time, and does not respond to changes in other objects.

### Design Criteria

A design criteria file (page 2472) and design check set (page 2472) can be applied to the profile if the parent alignment has a design speed applied. The criteria-based design feature enables you to verify that a profile design meets the minimum standards required by your local agency. These options are available if the Use Criteria-Based Design check box is selected.

**See also:**

■ Criteria-Based Profile Design (page 1044)

**To set profile properties**

**1** In Toolspace, on the Prospector tab, expand the Alignments collection, identify the parent alignment for the profile you want to work with, and then expand the Profiles collection for that alignment.

**2** Select the profile, right-click, and then click Properties. The Profile Properties (page 2174) dialog box opens.

**3** After setting properties, click OK.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Profile Properties

Toolspace Shortcut Menu

Prospector tab: Alignments ➤ <alignment-name> ➤ Profiles ➤ <profile-name> ➤ Properties

Object Shortcut Menu

Click profile ➤ Profile Properties

Command Line

EditProfileProperties

Dialog Box

Profile Properties Dialog Box (page 2174)

# Setting Profile View Properties

Set properties of profile views either from the Toolspace Prospector tab, or from the drawing window.

You can change the station range, grid, displayed profiles, and annotation of objects in the profile view.

Profile view properties are organized on several tabs of the Profile View Properties (page 2178) dialog box: information, stations, elevations, profiles, bands, hatch, projections, and pipe networks.

**Information tab**

View or edit the profile view name, description, and object style.

**Stations tab**

Set the range of stations displayed in the profile view.

**Elevations tab**

Set the range of elevations displayed in the profile view. If the profile view is split, you can use this tab to specify the stations and styles for the split segments. See The Profile View Object (page 1021) for more information about split profile views.

**Profiles tab**

Specify which profiles to display in the profile view and whether the view is clipped to the extents of a specific profile. You can edit the description, change the update mode (dynamic or static), layer, and style

for each profile in the current view. You can also specify whether to split the profile view, which reduces its vertical size.
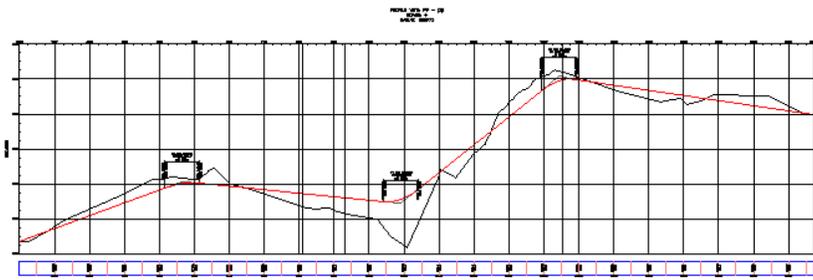


**Profile view without split**

The sample profile view before splitting shows an elevation range of 60 feet.
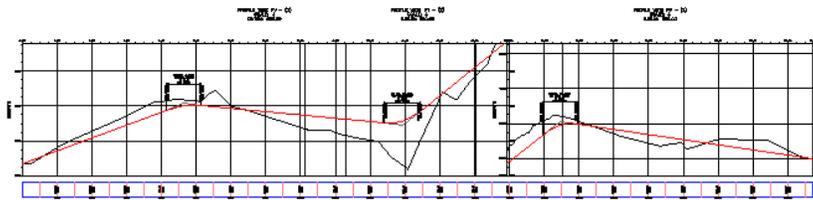


**Profile view with split**

The split profile view is more compact, with an elevation range of about 38 feet. At the split station, a separate elevation scale is provided for each part of the profile view.

### Bands tab

Specify which data bands to include in the profile view and whether they are placed along the top or bottom of the grid. The following data band types are available for displaying profile data: profile data, vertical geometry, horizontal geometry, superelevation for a specified horizontal alignment, sectional data, and pipe network data.

### Hatch tab

Specify the style and location of hatching between surfaces, which shows cut regions and fill regions in the vertical design.

### Pipe Networks tab

Specify which pipes or structures to display in the profile view as well as their styles, style overrides, and layer placement. The Pipe Networks tab is displayed only when there is one or more pipe networks in the drawing.

### Projection tab

Specify objects from plan view to project into the profile view, includings AutoCAD objects and AutoCAD Civil objects. You can also specify the object display styles, elevation options, and elevation values. The Projections tab is displayed only when projected objects (page 2480) exist in the profile view.

**See also:**

■ Setting Profile Properties (page 1041)

**To set profile view properties**

1   Click the profile view you want to modify.

2   Click Profile View tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚ .

3   In the Profile View Properties (page 2178) dialog box, set properties, and then click OK.

## Quick Reference

Ribbon

Click the profile view.  Click Profile View tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚

Toolspace Shortcut Menu

Prospector tab: Alignments ➤ <alignment name> ➤ Profile Views ➤ <profile view name> ➤ Properties

Object Shortcut Menu

Click profile view, right-click ➤ Profile View Properties

Command Line

EditGraphProperties

Dialog Box

Profile View Properties Dialog Box (page 2178)

# Criteria-Based Profile Design

Use the criteria-based design feature to apply agency-specific standards to a profile.

The criteria-based design feature provides the ability to verify that your profile design meets the minimum standards required by your local agency.

When you use the criteria-based design feature, you can select a design criteria file (page 2472), from which you can specify the minimum K value table. If the parent alignment also uses design criteria, the design criteria file applied to the alignment is applied for the profile by default. You can specify a separate design criteria file for the profile either during creation or in the profile properties.

When you lay out the profile, the appropriate minimum values specified in the design criteria file are displayed on the command line. You can either accept the default minimum value for a given sub-entity, or specify a new value.

Agency standards, which are typically based on superelevation and design speed requirements, are contained in the customizable design criteria file. When you create the profile, the minimum standards from the design criteria file ensure that the vertical curves comply with the minimum standards required by your local agency.

If the design parameters for a sub-entity violate the minimum values established in the design criteria file, a warning symbol appears both on the sub-entity in the drawing window and next to the violated value in the Profile Entities Vista (page 2166)and Profile Layout Parameters Dialog Box (page 2168). When you hover the cursor over a warning symbol, a tooltip displays which standard has been violated and how to correct the violation. The display of the warning symbol is controlled by the profile style.

Some profile design criteria are not available in table form in the design criteria file. For these criteria, you can define design checks (page 2472) to validate design standards. A design check must be saved in a design check set (page 2472) to be applied to a profile.

You can generate a report that documents validations or violations in the profile design. The design criteria report identifies whether each sub-entity within a given station range meets the appropriate design criteria and design checks.

**See also:**

■ Creating Layout Profiles (page 1048)

■ Applying Design Criteria to an Existing Layout Profile (page 1102)

■ Viewing and Correcting Profile Design Criteria Violations (page 1103)

## Profile Standards in the Design Criteria File

The design criteria file contains the standards tables that specify design standards for alignments and profiles.

The design criteria file (page 2472) contains A.A.S.H.T.O. (page 2467) standards tables for minimum K values at specified design speeds. Minimum K tables for Stopping Sight Distance, Passing Sight Distance, and Headlight Sight Distance are provided. If your local agency standards differ from the standards in the supplied design criteria file, you can use the Design Criteria File Editor dialog box (page 1843) to customize the file to support your local standards.

By default, the profile uses the design criteria applied to the parent alignment. You can accept the default, or select a different design criteria file for the profile.

The design criteria file contains standards for both alignment and profile objects. The Design Criteria Editor dialog box is used to modify the standards for both objects.

**NOTE** Design criteria files can be created only with the Design Criteria Editor dialog box. Existing design criteria data cannot be used unless it is input to an XML file using the Design Criteria Editor dialog box.

**See also:**

■ Design Criteria Files (page 867)

■ Creating a Design Criteria File (page 868)

■ Editing a Design Criteria File (page 870)

■ Criteria-Based Profile Design (page 1044)

■ Criteria-Based Alignment Design (page 866)

## Profile Design Checks

Create design checks to verify the design parameters for criteria that are not included in the design criteria tables.

The tables contained in the design criteria file (page 2472) contain A.A.S.H.T.O. (page 2467) standards tables for Stopping Sight Distance, Passing Sight Distance, and Headlight Sight Distance minimum K values at specified design speeds. Other criteria can be checked by using mathematical expressions, which are known as design checks (page 2472).

Design checks are created and managed in the same manner as label expressions. Use the Design Check Editor dialog box (page 1845), which is similar to the Expressions dialog box (page 1973), to create and edit design checks. To apply a design check to a profile, you must add it to a design check set (page 2472).

**NOTE** When you add curve design checks to a design check set, you must specify whether to apply it to sag curves, crest curves, or both.

Design checks differ from label expressions in that they always return a true or false value. For example, you can create a design check to issue a notification if a tangent is shorter than a specified minimum tangent length.

**See also:**

■ Design Check Examples (page 871)

■ Creating a Design Check (page 873)

■ Editing a Design Check (page 875)

■ Expressions (page 1547)

## Profile Design Check Sets

Save commonly used combinations of design checks in a design check set, which is then applied to a profile.

After you create a design check (page 2472), save it with an appropriate combination of other design checks in a design check set. A design check set can be applied to a profile either during the profile creation process, or through an existing profile's properties.

A design check set consists of design checks for a combination of design check types. For example, you can create a design check set that consists of separate design checks for minimum and maximum K values for sag and crest curves.

Design checks are created and managed in the same manner as label expressions. Use the Design Check Editor dialog box (page 1845), which is similar to the Expressions dialog box (page 1973), to create and edit design checks. To apply a design check to a profile, you must add it to a design check set (page 2472).

**See also:**

■ Profile Design Checks (page 1045)

■ Design Check Examples (page 871)

■ Creating a Design Check Set (page 875)

■ Editing a Design Check Set (page 876)

# Creating Profiles

Using profiles, you can view changes in elevation along a horizontal alignment.

In addition to the centerline profile, you can create offset profiles for features such as road edges or ditches.

Create a profile view to see one or more of the profiles associated with a horizontal alignment. Use the profile view of a surface as a guide for creating the layout profile of a designed surface, such as a road. On a profile view, you can also superimpose the profile of a different horizontal alignment that is in the same area.

To work most effectively with profiles, learn how to combine the create, copy, edit, and superimpose operations to get the profiles you need for terrain analysis and design.

In addition, you can use quick profiles to view and evaluate elevation information at locations where there is no alignment. Quick profiles are temporary objects that are useful for evaluating elevation information along a line, polyline, feature or lot line, survey figure, or along a series of points you select.

# Creating Surface Profiles

You can create surface profiles to show surface elevations along an alignment.

You can create a surface profile either from a horizontal alignment on a surface or from a surface profile file.

When you create a centerline profile, you can create one or more offset profiles at the same time. Use the Draw in Profile View option to display the profile in the drawing.
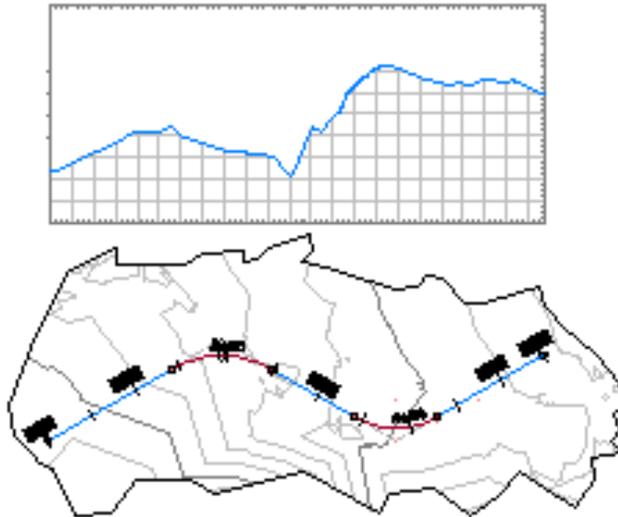


**To create profiles from a surface**

1 Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Surface Profile ⬚ .

2 In the Create Profile from Surface dialog box (page 2144), in the Alignment list, select the horizontal alignment to use for the profile.

3 Ensure that the surface you want to use is listed in the Select Surfaces list. If multiple surfaces are listed, use Ctrl + click to select one or more of them from which to create profiles.

4 Optionally, in the Station Range area, specify start and end stations if you want the profile to cover a distance less than the entire length of the horizontal alignment.

5 Click Add to add the centerline profile to the Profile List.

6 Optionally, select the Sample Offsets check box. Enter the offset distances in the field if you want the profile to include one or more offsets from the centerline of the horizontal alignment. Use positive numbers to specify right offsets, and negative numbers for left offsets. Use commas to separate the values. Example: 20,-20,30,40

7 Optionally, click Add to add the offset profiles to the Profile List.

8 In the table, review the data for each profile line and make corrections as required. Optionally, add or delete profiles in the Profile list.

9 Do one of the following:
   ■ If you want to draw the profile in a profile view, click Draw In Profile View, and then go to the create profile view procedure (page 1108).

■ Click OK. The profiles you created are saved. If the alignment has one or more profile views associated with it, the profiles you created are added to the views.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Surface Profile 

Menu

Profiles menu ➤ Create Profile From Surface

Command Line

CreateProfileFromSurface

Dialog Box

Create Profile from Surface (page 2144)

# Creating Layout Profiles

Create layout profiles to represent designed elevations along an alignment.

You typically draw a layout profile on the grid of a profile view, using the surface profile as a guide.

A profile view grid must exist in the drawing to use this command. Create curves and tangents by freehand sketching or by entering numeric values.

You can create a layout profile with tangents, and then optionally add curves at the points of vertical intersection (PVI).

You can create a profile from an external ASCII file that contains a series of stations along an alignment, the elevation of each station, and optionally, the length of curve at the station.

Curve attributes are set by the Curve Settings command. These curves maintain tangency when edited.

Several methods are available for drawing a layout profile:

■ Use the  Draw Tangents command to specify points of vertical intersection (PVIs) for a series of straight tangents. Later you can add free curves (page 1084) with specific parameters between the tangents.

■ Use the  Draw Tangents With Curves command to specify PVIs for the tangents, automatically creating curves between the tangents using the parameters you specify in the Vertical Curve Settings Dialog Box (page 2194).

■ Use the constraint-based profile design commands (page 1059) to create a profile one sub-entity at a time.

When you are prompted for point locations, you can use the Transparent Commands (page 1581) to specify points in the profile.

To create a more detailed design using a specific parameter or other constraints, or to create curves without first creating tangents, use the constraint-based tools included on the Profile Layout Tools toolbar. For more information, see Adding Tangents and Curves to a Layout Profile (page 1059).

The criteria-based design (page 2471) feature allows a layout profile to be validated against local design standards. You can select a design criteria file (page 2472) and/or a design check set (page 2472) using either the Create Profile - Draw New dialog box (page 2142) when you create the layout profile, or the Design Criteria tab (page 2175) of

the Profile Properties dialog box after it has been created. For more information, see Criteria-Based Profile Design (page 1044).

**To create a layout profile**

1   Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools ⬚ .

2   Click the profile view in which to draw the layout profile.

3   In the Create Profile - Draw New dialog box (page 2142), enter a unique Name for the profile. Optionally, enter a Description.

4   On the General tab (page 2142), specify the following settings:

   ■   Profile Style

   ■   Profile Layer

   ■   Profile Label Set

   ---

   **NOTE** If you do not want to apply design criteria to the profile, proceed to step 6.

   ---

5   If you would like to associate design standards with the profile, on the Design Criteria tab (page 2143), select the Use Criteria-Based Design check box.

   For more information, see Criteria-Based Profile Design (page 1044).

   On the Design Criteria tab, specify the following settings:

   ■   Use Design Criteria File
      Select this check box to associate a design criteria file with the profile. The default design criteria file location and the Default Criteria properties becomes available when you select the check box.

   ■   Use Design Check Set
      Select this check box to associate a design check set with the profile. The design check set list becomes available when you select the check box. Select a design check set from the list.

6   Click OK to display the Profile Layout Tools toolbar.

7   Use the commands on the Profile Layout Tools (page 2170) toolbar to draw the profile.

   You can use any of the following methods to lay out a profile:

   ■   Use the ⬚ Draw Tangents command to specify points of vertical intersection (PVIs) for the tangents, and then add free curves (page 1084) with specific parameters between the tangents.

   ■   Use the ⬚ Draw Tangents With Curves command to specify PVIs for the tangents, automatically creating curves between the tangents using the parameters you specify in the Vertical Curve Settings dialog box (page 2194).

   ■   Use various Profile Layout Tools (page 2170) to create separate entities and then join them together.

   ---

   **TIP** When you are prompted for point locations, you can use the Transparent Commands (page 1581) to specify points in the profile.

   ---

   When you use Use the ⬚ Draw Tangents With Curves command, curves are created to comply with the values in one of the following places:

   ■   If the profile has design criteria (page 866) applied to it, the minimum K value for the current design speed is applied to the curve. If no solution can be found with the minimum K value, the curve is not drawn.

■ If the profile does not have design criteria applied to it, the default curve length or K value specified in the vertical curve settings is applied to the curve. For more information, see Vertical Curve Settings dialog box (page 2194).

**To create a profile from a profile file**

1 Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Profile From File 🔲.

2 Browse to the text file (a text file uses the *.txt* extension).

3 Click the text file and click Open. If the file is correctly formatted, it is read into the system and the Create Profile - Draw New dialog box is displayed.

4 In the Create Profile - Draw New dialog box (page 2142), review and revise any settings you want to change for the profile name, description, and styles.

5 Click OK to save the profile in the drawing.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools 🔲

Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Profile From File 🔲

Menu

Click Profiles  menu ➤ Create Profile By Layout
Click Profiles  menu ➤ Create Profile From File

Command Line

CreateProfileLayout
CreateProfileFromFile

Dialog Box

Create Profile - Draw New (page 2142)
Profile Layout Tools (page 2170)

## Drawing Tangent-Tangent Profiles

Use the Profile Layout Tools to create and edit profiles.

You can use either of the following commands to create a quick layout of the profile. The lines and curves you create maintain tangency when you edit them.

**Draw Tangents Without Curves**

Use this command to create a quick layout of the profile. Specify tangent start and end points (1 through 4), then add curves at the points of intersection as necessary. The tangents are fixed, so you can edit them and they always maintain tangency.
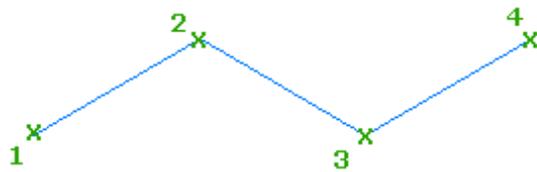
**Draw Tangents With Curves**

Use this command to create a quick layout of the profile, with curves automatically created at points of intersection. Specify tangent start and end points (1 through 4), and curves are created at the points of intersection (2, 3) based on the values you enter in the Curve Settings dialog box. You can edit the tangents and curves and they always maintain tangency.



**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

**See also:**

■ Editing PVIs (page 1093)

**To draw a profile using the tangent-tangent method with no curves**

1  Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools .

2  In the Create Profile-Layout (page 1763) dialog box, enter creation information.

3  On the Profile Layout Tools toolbar, click  Draw Tangents Without Curves.

4  Specify the start point.

5  Continue to specify points. Press Enter to end the command, or click another command from the Profile Layout Tools toolbar.

**To draw a profile using the tangents with curves**

1  Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools .

**2** In the Create Profile-Layout (page 1763) dialog box, enter creation information.

**3** On the Profile Layout Tools toolbar, click ⛤ Draw Tangents With Curves.

**4** Specify the start point.

**5** Continue to specify points. Press Enter to end the command or select another command from the Profile Layout Tools.

Curves are created to comply with the values in one of the following places:

■ If the profile has design criteria (page 866) applied to it, the minimum radius for the current design speed is applied to the curve. If no solution can be found with the minimum radius value, the curve is not drawn.

■ If the profile does not have design criteria applied to it, the default value specified in the curve and spiral settings is applied to the curve. For more information, see Curve and Spiral Settings (page 861).

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools 📈

Menu

Click Profiles menu ➤ Create Profile By Layout

Command Line

CreateProfileLayout

Dialog Box

Create Alignment - Layout Dialog Box (page 1763)
Profile Layout Tools (page 2170)

## Setting Curve Parameters

Enter the curve settings you want to use when creating profiles or parcels using the Draw Tangents With Curves command.

You can specify default settings for curves by editing the Profile Creation settings in the CreateProfileLayout command.

## Specifying Curve Settings

Specify default curve types and values in the Vertical Curve Settings dialog box or by editing the Profile Creation settings for the CreateProfileLayout command.

The parameter values in the Vertical Curve Settings dialog box apply only to the curves created using the Draw Tangents with Curves command in the current profile. You can specify default settings for all curves by editing the settings in the CreateProfileLayout command.

**To change the default CurveSettings**

**1** On the Settings tab in Toolspace, expand the Profiles collection and then expand the Commands collection.

**2** Right-click CreateProfileLayout ➤ Edit Command Settings.

**3** In the Edit Command Settings-CreateProfileLayout dialog box, expand the Profile Creation property.

**4** Specify the settings, click Apply, and click OK.

> **NOTE** To save these command settings at the drawing level, on the Ambient Setting tab (page 68)select Yes in the value column for Save Command Changes To Settings.

**To change the curve settings when you draw a profile using the tangent with curves method**

**1** Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools .

**2** In the Create Profile - Draw New (page 2142) dialog box, enter creation information.

**3** On the Profile Layout Tools toolbar, click Curve Settings.

**4** In the Vertical Curve Settings (page 2194) dialog box, enter curves values.

> **NOTE** The curve type you specify in the Vertical Curve Settings dialog box is used in all curve commands on the Profile Layout Tools toolbar.

### Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Profile Creation Tools 

Menu

Click Profiles menu ➤ Create Profile By Layout

Command Line

CreateProfileLayout

Profile Layout Tools Toolbar Icon

Curve Settings

Dialog Box

Vertical Curve Settings Dialog Box (page 2194)

# Format Requirements of Profile Files

Use profile files to record layout data for points of vertical intersection (PVIs) in a profile.

A profile file is a simple text file in ASCII format. The file contains a series of stations along an alignment, the elevation of each station, and optionally, the length of curve at the station. Each line can include up to three information elements, separated by spaces:

**1** **Station**. An element required for each line. Plus characters (+) cannot be used.

**2** **Elevation**. A required element.

**3** **Curve length**. An optional element. However it cannot be used on the first or last line of the file. All vertical curves must be parabolic.

Sample file lines:

```
0 100.23
50 104.2
100 109.0 40.0
150 102.63
200 98.12 35.0
250 102.45
300 105.8
```

In this example, only the third and fifth PVIs include curves (lengths of 40.0 and 35.0). Use these other format requirements for each profile file:

■ The first and last lines must not contain curve information.

■ The first line must provide the station and elevation for the first station.

■ Stations must be in ascending order.

■ No leading blank lines or headers.

■ No blank lines; a blank line is read as the end of the file.

■ No blank spaces at the beginning of any line.

■ The last item must be the elevation for the last station.

## Creating a Superimposed Profile

In a profile view, display the profile of another alignment that is adjacent to the main horizontal alignment.

Use this option to show one profile in relation to another, such as a ditch in relation to a road centerline.

The general process is first to create separate profile views to display the two profiles. Then identify one profile as the source profile to display in the other profile view, which is known as the destination profile view. In the example of a ditch and a road centerline, let us assume that the ditch is the source profile and the profile view for the road is the destination. When you superimpose the ditch profile on the profile view of the road, you see the ditch elevations with reference to the stations along the road centerline.

When you superimpose a profile, you can set some options in the Superimpose Profile Options dialog box (page 2193)

**To superimpose a profile**

1 Identify the source profile you want to superimpose. Ensure that each profile appears in a profile view.

2 Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Superimposed Profile 📉 .

3 In the drawing, click the source profile line.

4 Click the grid of the destination profile view.

5 Optionally, on the Limits tab (page 2194), specify a limited range of stations to superimpose.

**NOTE** If you want the superimposed profile station range to remain fixed, then select the Select Start and Select End checkboxes and the superimposed profile will not react to changes in the station range of the source alignment. If the Select Start and Select End boxes are not selected, then the start and end of the superimposed profile locks to the start and end of the source alignment. If the start or end of the source alignment changes, the superimposed profile updates dynamically.

6 Optionally, on the Accuracy tab (page 2194), specify different mid-ordinate distances for curve tessellation.

**7** Click OK.

### Quick Reference

# Creating a Corridor Profile

In a profile view, display a profile derived from a corridor feature line.

Profiles derived from corridor feature lines are static and are displayed with a ⚲ in the Profile collection in the Prospector tree.

For information on creating a corridor profile, see Exporting Corridor Feature Lines as Profiles (page 1372).

# Creating a Profile by Reference

Create a reference to an existing profile in a data shortcut (page 2472) or a Vault project.

The profile reference is a lightweight read-only copy of the original, but it gives you access to profile data for other objects. Before you create a profile reference in the current drawing, the reference object or data shortcut must exist in the current project collection on the Toolspace Prospector tab.

When a profile reference is created, a reference to the profile's parent alignment is also created in the drawing. AutoCAD Civil automatically looks for the correct alignment reference in the source drawing when creating the profile reference.

**See also:**

■ Creating Data Shortcuts (page 131)

■ Using Vault (page 138)

**To create a reference to a data shortcut profile**

**1** In Toolspace, on the Prospector tab, ensure that the correct project folder is identified on the Data Shortcuts node.

**2** Expand the Data Shortcuts ➤ Alignments collection.

**3** Expand the parent alignment of the profile you want to reference.

**4** Right-click the desired profile, and then click Create Reference.

The Create Profile Reference dialog box is displayed, in which you can optionally change the properties of the reference profile, as described in the following steps.

**5** In the Create Profile Reference dialog box, change the source of the reference by selecting a source in the Source Profile list.

**6** Change the source parent alignment by selecting an alignment from the Source Alignment list.

**7** Enter a Name for the profile.

> **NOTE** To name the profile, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**8** Specify a Profile Style or accept the default style.
For more information, see Profile Styles (page 1027).

**9** Click  to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the profile is placed on the current layer.

**10** Specify a Profile Label Set or accept the default label set.

**11** Click OK to create the profile reference.
A reference to the parent alignment is also created. The alignment name is displayed under the

Alignments collection in the Prospector tree with a  next to it. Expand the alignment name and the Profiles collection to view the profile reference.

**To create a reference to a profile in a Vault project**

**1** In Toolspace, on the Prospector tab, in Master View, expand the Projects <project name> ➤ Alignments collection.

**2** Expand the parent alignment of the profile you want to reference.

**3** Right-click the desired profile, and then click Create Reference.
The Create Profile Reference dialog box is displayed, in which you can optionally change the properties of the reference profile, as described in the following steps.

**4** In the Create Profile Reference dialog box, change the source of the reference by selecting a source in the Source Profile list.

**5** Change the source parent alignment by selecting an alignment from the Source Alignment list.

**6** Enter a Name for the profile.

> **NOTE** To name the profile, click its default name. Enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

**7** Specify a Profile Style or accept the default style.
For more information, see Profile Styles (page 1027).

**8** Click  to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the profile is placed on the current layer.

**9** Specify a Profile Label Set or accept the default label set.

**10** Click OK to create the profile reference.

A reference to the parent alignment is also created. The alignment name is displayed under the Alignments collection in the Prospector tree with a ↗ next to it. Expand the alignment name and the Profiles collection to view the profile reference.

### Quick Reference

Toolspace Right-Click Menu (for data shortcut project)

Prospector tab: Data Shortcuts ➤ Alignments ➤ <alignment name> ➤ Profiles ➤ Create Reference

Toolspace Right-Click Menu (for Vault project)

Prospector tab: Projects ➤ <project name> ➤ Alignments ➤ <alignment name> Profiles ➤ Create Reference

Command Line

CreateProfileReference

Dialog Box

Create Profile Reference (page 2201)

# Copying a Profile

Copy a layout profile or a static surface profile. Use the copy to create a profile or replace an existing one.

Profile copying operations allow you to create a profile that is based on another. For example, you can create an offset profile for a road edge or ditch that is a specific elevation below the road centerline. In this case, you can copy the centerline profile, then change the elevation of the copy to create the offset profile.

After copying a profile, you can raise or lower the copy to create an initial line for a related profile. For example, to place a ditch five feet lower than a centerline, copy the centerline profile and lower the copy five feet to create a ditch profile.

### Troubleshooting a Copied Profile

When you try copying a profile, you may receive the following error message:

*Station xx+xx: Either outside of alignment limits, or within gap created by a station equation.*

To resolve this issue, create a profile that extends beyond the station limits of the defined parent alignment. You will not be able to copy a selected station range of PVIs if you select a start or end station that is outside the limits of the parent horizontal alignment. Instead, copy the entire profile and then remove the unwanted PVIs manually.

### To copy a profile

1 Click the profile you want to copy.

2 Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

3 In the Profile Layout Tools dialog box (page 2170), click  Copy Profile.

4 In the Copy Profile Data Dialog Box (page 2141), check the data displayed in the Source Profile Information area to ensure that this is the correct profile to copy.

5 Click the PVI range you want to copy: either All PVIs or those within a range of stations.

6 If you clicked Station Range, specify the start and end stations.

7   In the Destination Profile Options area, specify what to do with the copy of the profile: either Create New Profile or Overwrite Existing Profile.

8   If you clicked Overwrite Existing Profile, select the profile to overwrite.

9   To overwrite all properties of the existing profile, select Overwrite All Properties.

10  Click OK to create the copy of the profile. The copy is initially named according to the naming template, like any new profile. You can change the name. The copy is displayed in the current profile view, superimposed on its source profile. It also appears in the Toolspace Prospector tab, in the list of profiles for the parent alignment.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Dialog Box

Copy Profile Data Dialog Box (page 2141)

## Creating a Quick Profile

View surface elevation data along an object or between two points.

Use the quick profile functionality to generate elevation data using a few quick steps before you create a detailed design.

You can create quick profiles along many object types:

■   2D or 3D lines or polylines

■   Lot lines

■   Feature lines

■   Survey figures

■   A series of points

These profiles are not saved with the drawing.

You can control which surfaces are sampled in the quick profile, the style used to display the profile of each surface, and the profile view style.

To create a profile that you can use again, create an alignment at the location of interest and use it to generate a standard profile.

After you create a quick profile, extract information from it using the Inquiry tool. You can select and move a quick profile. Use the object shortcut menu to change the style of one of its profiles or the profile view.

**NOTE** To simplify working with quick profiles, open two viewports before you create a quick profile. Use one viewport to display the plan view and the other viewport to display the quick profiles.

**To create a quick profile from an object**

**1** Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Quick Profile  .

On the command line, you are prompted to pick an object.

**2** In the drawing, select a line, polyline, lot line, feature line, or survey figure.

**3** In the Create Quick Profiles dialog box, select the surfaces you want to sample, and the profile and profile view styles.

**4** Optionally, if you selected a 3D object, select Draw 3D Entity Profile to display elevation data for the object in the quick profile and a profile style for the 3D object elevation data.

**5** Click OK.

**6** In the drawing window, click a location for the lower left corner (origin) of the profile view grid.

The quick profile is drawn.

**To create a quick profile by selecting points**

**1** Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Quick Profile  .

On the command line, you are prompted to pick an object.

**2** Enter p (points).

**3** In the drawing, select a point, and then a second point.

**4** Optionally, select additional points as required.

**5** Press Enter.

**6** In the Create Quick Profiles dialog box, select the surfaces you want to sample, and the profile and profile view styles.

**7** In the drawing window, click a location for the lower left corner (origin) of the profile view grid.

The quick profile is drawn.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Quick Profile 

Menu

Click Profiles menu ➤ Quick Profile

Command Line

CreateQuickProfile

Dialog Box

Create Quick Profiles Dialog Box (page 2153)

# Adding Tangents and Curves to a Layout Profile

You can add fixed, floating, or free tangents and curves to a layout profile.

Use constraint-based commands on the Profile Layout Tools (page 2170) toolbar to refine the layout profile design to meet your project objectives.

**NOTE** When drawing or editing constraint-based profile entities, ensure that the Profile Layout Tools (page 2170) toolbar is set to △╱ Entity Based mode.

The three entity type describes the way an entity is defined and edited:

■ Fixed entities are defined by specifying parameters, such as pass-through points, tangent slope, and curve radius. The entities are fixed in position and must be edited directly. Except for its length, the geometry of a fixed entity is not dependent on adjacent entities. If an adjacent entity is edited, the geometry of the fixed entity is not affected.

■ Floating entities are defined by specifying parameters and an entity to attach to. Floating entities are always tangent to the entity they are attached to, and are dependent on the other entity to define their geometry.

■ Free entities are defined by attachment to two entities, before and after. The free entity always remains tangent to the two adjacent entities, and its geometry is defined by these adjacent entities.

# Adding Vertical Tangents

Add fixed, free, or floating vertical tangents to build constraint-based profile geometry.

To access the vertical tangent creation commands, click the arrow next to the tangent creation command displayed on the toolbar. The ╱ Draw Fixed Tangent By Two Points command is displayed by default.

After clicking a button on the Profile Layout Tools toolbar, follow the command prompts on the command line.

**TIP** You can use transparent commands to specify a station and elevation when you are prompted to select a point. For more information, see Transparent Commands (page 1581).

## Adding Fixed Vertical Tangents

Add fixed vertical tangents to build constraint-based profile geometry.

Fixed entities are defined by specifying parameters, such as points or minimum radius. The parameters are fixed in their position and must be edited directly.

The position of a fixed entity is not affected by edits to adjacent entities.

Except for its length, the geometry of a fixed entity is not dependent on adjacent entities.

## Adding a Fixed Vertical Tangent with Two Points

Add a fixed vertical tangent by specifying two points that the line will pass through.

**To add a fixed vertical tangent with two points**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉 .

2 On the Profile Layout Tools (page 2170) toolbar, click ╱ Draw Fixed Tangent By Two Points.

3 Specify the start point.

**4** Specify the next point.

## Quick Reference

Ribbon

Click profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

Draw Fixed Tangent By Two Points

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Tangent by Best Fit

Add a fixed, two-point vertical tangent entity by best fit to a profile. You can define the best fit entity using a series of AutoCAD Civil points or clicks on screen.

The following illustration shows a fixed vertical tangent created by best fit. The Xs indicate the data points that were used to create the entity.



**NOTE** Each best fit profile entity is drawn to the vertical scale set in the profile view style. To create an accurate best fit entity, set the profile view style Vertical Exaggeration value to 1.000 to match the horizontal scale of the profile view. For more information, see Creating and Editing Profile View Styles (page 1030).

**See also:**

■ Editing Best Fit Profile Entities (page 1100)

**To add a fixed vertical tangent by best fit from AutoCAD Civil points**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**3** In the Profile Layout Tools (page 2170) toolbar, click  Fixed Tangent - Best Fit.

**4** In the Tangent by Best Fit dialog box (page 1989), select From COGO Points.

**5** In plan view, select two or more points. Enter G to select a point group or N to enter points by number.

As the points are selected in plan view, an X marks each regression point and a temporary, dashed tangent is displayed in the profile view.

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the tangent:

■ Click  to create the tangent and keep the Regression Data vista open.

■ Click  to create the tangent and close the Regression Data vista.

**To add a fixed vertical tangent by best fit by clicking on screen**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**3** In the Profile Layout Tools (page 2170) toolbar, click  Fixed Tangent - Best Fit.

**4** In the Tangent by Best Fit dialog box (page 1989), select By Clicking On The Screen.

**5** Select a starting point and at least one other point.

**NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed tangent is displayed in real time.

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the tangent:

■ Click  to create the tangent and keep the Regression Data vista open.

■ Click  to create the tangent and close the Regression Data vista.

**To add a fixed vertical tangent by best fit from existing entities**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

**3** In the Profile Layout Tools (page 2170) toolbar, click  Fixed Tangent - Best Fit.

**4** In the Tangent by Best Fit dialog box (page 1989), select By Clicking On The Screen. Specify the tessellation and mid-ordinate tolerance settings.

**5** Select one of the entities listed on the command line.

---

**NOTE** You can select several types of entities listed on the command line.

---

**6** If you selected a profile entity, specify the starting and ending station on the Specify Station Range dialog box (page 1992).

**7** Repeat steps 5 and 6 to select other entities from which to create the best fit tangent.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed tangent is displayed in real time.

**8** Press Enter to complete the command.

**9** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**10** Create the tangent:

■ Click  to create the tangent and keep the Regression Data vista open.

■ Click  to create the tangent and close the Regression Data vista.


## Quick Reference

Ribbon

Click profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Fixed Tangent - Best Fit

Dialog Box

Profile Layout Tools (page 2170)
Tangent by Best Fit (page 1989)
Regression Data (page 1990)


## Adding Floating Vertical Tangents

Add floating vertical tangents to build constraint-based profile geometry.

Floating entities are defined by specifying parameters and an entity to attach to. A floating entity is always tangent to the attached entity and is dependent on the other entity to define its geometry.

## Adding a Floating Vertical Tangent with a Pass Through Point

Add a floating vertical tangent by specifying an attachment entity and a pass-through point.

The figure shows a floating tangent from an attached entity (1) to a pass-through point (2).
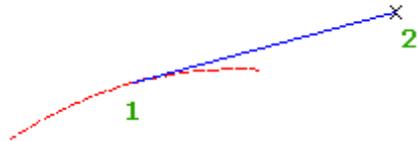


**To add a floating vertical tangent with a specified pass-through point**

1  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✎ .

2  On the Profile Layout Tools toolbar, click ⚲ Float Tangent (Through Point).

3  Specify the fixed or floating curve to which you want to attach the tangent.

4  Specify the pass-through point.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✎

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

⚲ Float Tangent (Through Point)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Floating Vertical Tangent by Best Fit

You can define the best fit entity using a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

The illustration shows a floating vertical tangent created by best fit from an existing endpoint (1). The Xs indicate the data points used to create the entity.



**NOTE** Each best fit profile entity is drawn to the vertical scale set in the profile view style. To create an accurate best fit entity, set the profile view style Vertical Exaggeration value to 1.000 to match the horizontal scale of the profile view. For more information, see Creating and Editing Profile View Styles (page 1030).

**See also:**

■ Editing Best Fit Profile Entities (page 1100)

**To add a floating vertical tangent by best fit from AutoCAD Civil points**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

3 In the Profile Layout Tools (page 2170) toolbar, click  Float Tangent - Best Fit.

4 Select an existing fixed or floating profile entity to which to attach the best fit tangent.

5 In the Tangent by Best Fit dialog box (page 1989), select From COGO Points.

6 In plan view, select two or more points. Enter G to select a point group or N to enter points by number.

   As the points are selected in plan view, an X marks each regression point and a temporary, dashed tangent is displayed in the profile view.

7 Press Enter to complete the command.

8 In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

   As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

9 Create the tangent:

   ■ Click  to create the tangent and keep the Regression Data vista open.

   ■ Click  to create the tangent and close the Regression Data vista.

**To add a floating vertical tangent by best fit by clicking on screen**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**3** In the Profile Layout Tools (page 2170) toolbar, click Float Tangent - Best Fit.

**4** Select an existing fixed or floating profile entity to which to attach the best fit tangent.

**5** In the Tangent by Best Fit dialog box (page 1989), select By Clicking On The Screen.

**6** Select at least one point in the drawing.

> **NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed tangent is displayed in real time.

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the tangent:

- Click to create the tangent and keep the Regression Data vista open.

- Click to create the tangent and close the Regression Data vista.

**To add a floating vertical tangent by best fit from existing entities**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**3** In the Profile Layout Tools (page 2170) toolbar, click Float Tangent - Best Fit.

**4** Select an existing fixed or floating profile entity to which to attach the best fit tangent.

**5** In the Tangent by Best Fit dialog box (page 1989), select From Entities. Specify the tessellation and mid-ordinate tolerance settings.

**6** Select one of the entities listed on the command line.

> **NOTE** You can select several types of entities listed on the command line.

**7** If you selected a profile entity, specify the starting and ending station on the Specify Station Range dialog box (page 1992).

**8** Repeat steps 6 and 7 to select other entities from which to create the best fit tangent.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed tangent is displayed in real time.

**9** Press Enter to complete the command.

**10** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**11** Create the tangent:

■ Click 🖫 to create the tangent and keep the Regression Data vista open.

■ Click ✔ to create the tangent and close the Regression Data vista.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉

Menu

Click Profiles menu ➤ Edit Profile Geometry.

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

🔏 Float Tangent - Best Fit

Dialog Box

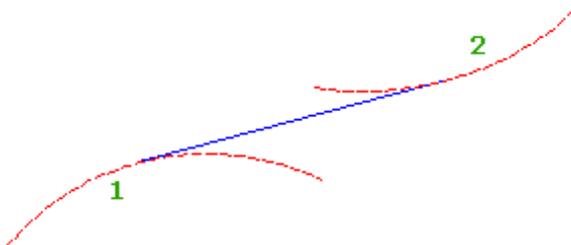Profile Layout Tools (page 2170)
Tangent by Best Fit (page 1989)
Regression Data (page 1990)

## Adding Free Vertical Tangents

Add free vertical tangents to build constraint-based profile geometry.

A free entity is defined by attachment to two entities, which define the geometry of the free entity.

The illustration shows a free tangent between two curves.

**To add a free vertical tangent between two curves**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

2   On the Profile Layout Tools toolbar, click  Free Tangent.

3   Specify the first fixed or floating curve to which you want to attach the tangent.

4   Specify the second fixed or floating curve to which you want to attach the tangent.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Free Tangent

Dialog Box

Profile Layout Tools (page 2170)

# Adding Vertical Curves

Add fixed, free, or floating curves to build constraint-based profile geometry.

To access the vertical curve creation commands, click the arrow next to the curve creation command displayed

on the toolbar. The  Fixed Vertical Curve (Three Points) command is displayed by default.

After clicking a button on the Profile Layout Tools toolbar, follow the command prompts on the command line.

---

**TIP** You can use transparent commands to specify a station and elevation when you are prompted to select a point. For more information, see Transparent Commands (page 1581).

---

## Vertical Curve Design

Designing vertical curves in layout profiles requires that you understand the concepts described in this topic.

**Curve Types**

- **Taylor Pohlman Parabolic**. Most vertical curves in road design are symmetrical parabolic curves for a good reason. The parabolic curve is the natural vertical curve followed by any projectile. A properly designed symmetrical parabola minimizes the inertial forces on a vehicle traveling along the curve. Highway curves are often designed with reference to curve tables, such as those provided by the American Association of State Highway and Transportation Officials (A.A.S.H.T.O. (page 2467)). These tables can be used only with symmetrical parabolic curves.

Quadratic parabolas are preferred in vertical designs as they have a constant rate of change of grade along the curve. Quadratic parabolas are a very close approximation to circular curves at flat grades and are usually used in vertical design. This quality makes them very easy to stake out. Substituting these values we get the quadratic parabola equation for vertical curve as:

$$y = \frac{g_1}{100} x + \frac{(g_2 - g_1)}{200L} x^2$$

- g1 is the grade of tangent in

- g2 is the grade of tangent out

- L is the horizontal length of the curve that is fit between these two tangents

Since the rate of grade change is constant over the length of the curve, the parabolic curve parameter can be expressed as:

$$K = \frac{L}{|g_1 - g_2|}$$

- Circular. These curves are easier to lay out and construct. This type is often used for the design of railways or roads that are traveled at low speeds.

- Asymmetrical parabolic. These curves are the least common type. This curve type consists of two different parabolic curves that meet at the midpoint of the vertical curve.

**K-Value**

This value represents the horizontal distance along which a 1% change in grade occurs on the vertical curve. It expresses the abruptness of the grade change in a single value. Speed tables or other design tools often provide a target minimum K value.

The criteria-based design (page 1044) feature automatically validates profile curve design against local design standards. A.A.S.H.T.O. tables defining minimum K values at given speeds are provided in the customizable design criteria file. Curves that violate your design standards can be corrected by using the Profile Layout Parameters dialog box (page 2168) to either adjust K value directly, or change curve length and tangent grade to meet the criteria.

**K-value formula**
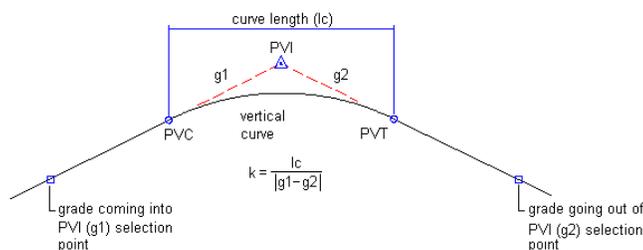


**Stopping Sight Distance**

This design method for crest curves provides a minimum curve length. The curve must be long enough so that the driver of a standard vehicle can always see an object before it gets within the maximum stopping distance for the designed speed of travel.

### Diagram of stopping sight distance



### Passing Sight Distance

This design method for crest curves provides a minimum curve length. The curve must be long enough so that the driver of a standard vehicle can always see an oncoming vehicle within a safe distance for the designed speed of travel.

### Diagram of passing sight distance
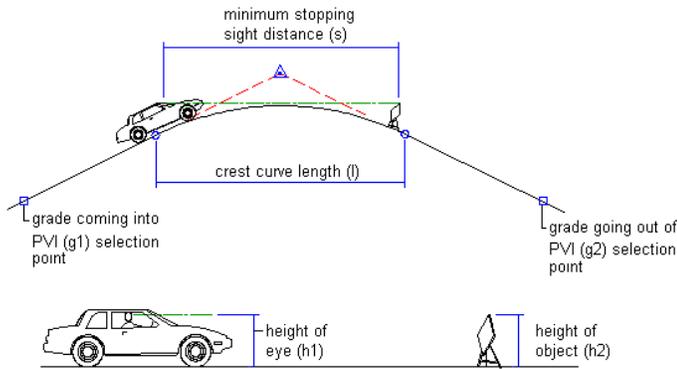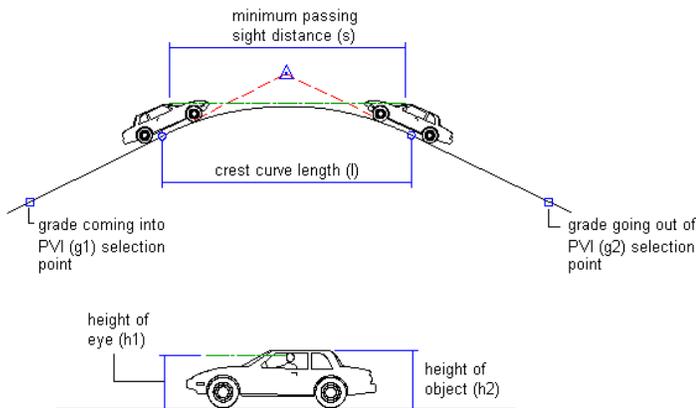


### Headlight Sight Distance

This design method for sag curves provides a minimum curve length. The curve must be long enough so that in dark driving conditions, the headlights of a standard vehicle illuminate the road a safe distance beyond the stopping distance for the designed speed of travel.

**Diagram of headlight sight distance**
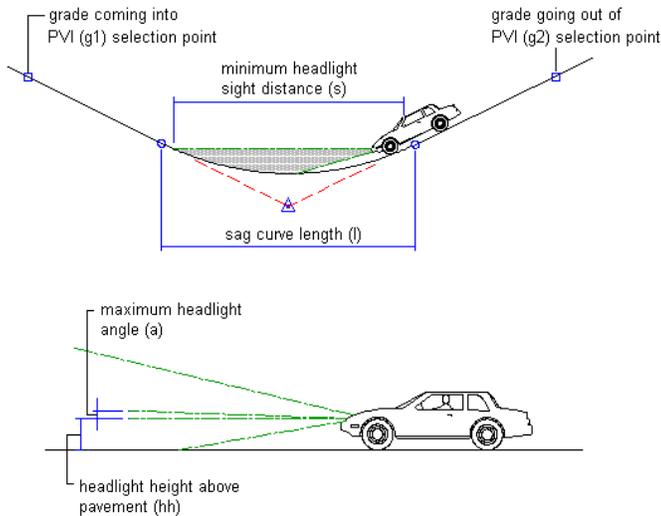


**Rider Comfort**

This design method for sag curves provides a minimum curve length. The curve must be long enough so that people in a standard vehicle do not experience excessive inertial force as they travel through the curve at the designed speed.

**Diagram of rider comfort design**
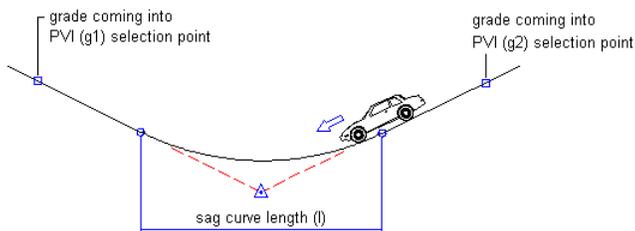


# Adding Fixed Vertical Curves

Add fixed vertical curves to build constraint-based profile geometry.

Fixed entities are defined by specifying parameters, such as points or minimum radius. The parameters are fixed in their position and must be edited directly. If an adjacent entity is edited, the geometry of the fixed entity is not affected. Except for its length, the geometry of a fixed entity is not dependent on adjacent entities.

# Adding a Fixed Vertical Curve with Three Points

Add a fixed vertical curve by specifying three pass-through points.

The illustration shows a fixed curve from start point 1 through specified points 2 and 3.
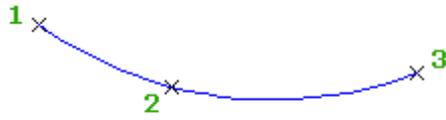
**To add a fixed vertical curve with three points**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon].

2 On the Profile Layout Tools toolbar, click ![icon] Fixed Vertical Curve (Three Points).

3 Specify the start point.

4 Specify the second point.

5 Specify the end point.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon]

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

![icon] Fixed Vertical Curve (Three Points)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Curve with Two Points and a Parameter

Specify two pass-through points and a K value or minimum radius.

The illustration shows a fixed curve defined by a start point (1), end point (2), and a minimum radius (3):
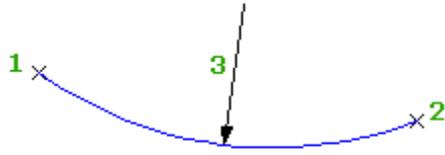
**To add a fixed vertical curve with two points and a parameter**

1　Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✎ .

2　On the Profile Layout Tools toolbar, click 🔔 Fixed Vertical Curve (Two Points, Parameter).

3　Specify a start point.

4　Specify an end point.

5　Do one of the following:
   - To specify a crest curve, press Enter.
   - To specify a sag curve, enter S (sag).

6　Do one of the following:
   - Specify a K value.
   - To specify a minimum radius for the curve parabola, enter R (radius), and then specify the radius.

     **NOTE** If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at given design speeds are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

## Quick Reference

Ribbon

　Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✎

Menu

　Click Profiles  menu ➤ Edit Profile Geometry

Command Line

　EditProfileLayout

Profile Layout Tools Toolbar Icon

　🔔 Fixed Vertical Curve (Two Points, Parameter)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Curve from an Entity with a Pass-Through Point

Add a fixed vertical curve by specifying an attachment entity and a pass-through point.

The illustration shows a fixed curve from an attached entity (1) and a pass-through point (2):
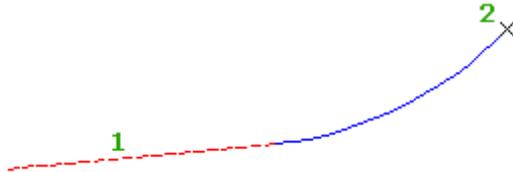


**To add a fixed vertical curve with an entity and a pass-through point**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   On the Profile Layout Tools toolbar, click  Fixed Vertical Curve (Entity End, Through Point).

3   Select the fixed or floating entity to which you want to attach the curve.

    The nearest end point of the entity is the start point of the curve. The grade of the start point is the start grade for the curve.

4   Specify a pass-through point for the curve.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Fixed Vertical Curve (Entity End, Through Point)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Curve with Two Points and a Start Point Grade

Add a fixed vertical curve by specifying two pass-through points and a grade at the start point.

The illustration shows a fixed curve from a start point (1) to an end point (2) and the grade at the start point (3):



**To add a fixed vertical curve with two points and the start point grade**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬚ .

2 On the Profile Layout Tools toolbar, click ⬚ Fixed Vertical Curve (Two Points, Grade At Start Point).

3 Specify a start point.

4 Specify an end point.

5 Enter the grade value at the start point.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬚

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

⬚ Fixed Vertical Curve (Two Points, Grade At Start Point)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Curve with Two Points and an End Point Grade

Add a fixed vertical curve by specifying two pass-through points and a grade at the end point.

The following illustration shows a fixed curve from a start point (1) to an end point (2) and the grade at the end point (3):



**To add a fixed vertical curve with two points and the end-point grade**

1  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬛✎ .

2  On the Profile Layout Tools toolbar, click ⬛ Fixed Vertical Curve (Two Points, Grade At End Point).

3  Specify a start point.

4  Specify an end point.

5  Enter the grade at the end point.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬛✎

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

⬛ Fixed Vertical Curve (Two Points, Grade At End Point)

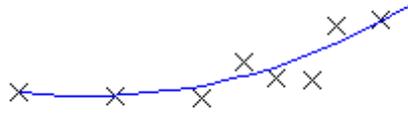Dialog Box

Profile Layout Tools (page 2170)

## Adding a Fixed Vertical Curve by Best Fit

Add a fixed, three-point curve entity by best fit to a profile. You can define the best fit entity using a series of AutoCAD Civil points, existing entities, or clicks on screen.

The illustration shows a fixed vertical curve created by best fit. The Xs indicate the data points used to create the entity.

**NOTE** Each best fit profile entity is drawn to the vertical scale set in the profile view style. To create an accurate best fit entity, set the profile view style Vertical Exaggeration value to 1.000 to match the horizontal scale of the profile view. For more information, see Creating and Editing Profile View Styles (page 1030).

**See also:**

■ Editing Best Fit Profile Entities (page 1100)

**To add a fixed vertical curve by best fit from AutoCAD Civil points**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉 .

3 In the Profile Layout Tools (page 2170) toolbar, click 🔲 Fixed Vertical Curve - Best Fit.

4 In the Curve by Best Fit dialog box (page 1989), select From COGO Points.

5 In plan view, select three or more points. Enter G to select a point group or N to enter points by number.

As the points are selected in plan view, an X marks each regression point and a temporary, dashed curve is displayed in the profile view.

6 Press Enter to complete the command.

7 In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

8 Create the curve:

■ Click 🖫 to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To add a fixed vertical curve by best fit by clicking on screen**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉 .

3 In the Profile Layout Tools (page 2170) toolbar, click 🔲 Fixed Vertical Curve - Best Fit.

**4** In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen.

**5** Select a starting point and at least two other points.

> **NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**6** Press Enter to complete the command.

**7** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**8** Create the curve:

- Click ⊟ to create the curve and keep the Regression Data vista open.

- Click ✔ to create the curve and close the Regression Data vista.

**To add a fixed vertical curve by best fit from existing entities**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**3** In the Profile Layout Tools (page 2170) toolbar, click  Fixed Vertical Curve - Best Fit.

**4** In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen. Specify the tessellation and mid-ordinate tolerance settings.

**5** Select one of the entities listed on the command line.

> **NOTE** You can select several types of entities listed on the command line.

**6** If you selected a profile entity, specify the starting and ending station on the Specify Station Range dialog box (page 1992).

**7** Repeat steps 5 and 6 to select other entities from which to create the best fit curve.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**8** Press Enter to complete the command.

**9** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**10** Create the curve:

- Click ⊟ to create the curve and keep the Regression Data vista open.

- Click ✔ to create the curve and close the Regression Data vista.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Fixed Vertical Curve - Best Fit

Dialog Box

Profile Layout Tools (page 2170)
Curve by Best Fit (page 1989)
Regression Data (page 1990)

# Adding Floating Vertical Curves

Add floating vertical curves to build constraint-based profile geometry.

Floating entities are defined by specifying parameters and an entity to attach to. Floating entities are always tangent to the entity it is attached to, and are dependent on the other entity to define its geometry.

# Adding a Floating Vertical Curve with a Pass-Through Point and a Parameter

Add a floating vertical curve by specifying a pass-through point and a parameter.

The illustration shows a floating curve from an existing entity (1) to a pass-through point (2) and a minimum radius (3). This curve type can also be drawn from a specified K value.



**To add a floating vertical curve with a pass-through point and a parameter**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

2   On the Profile Layout Tools toolbar, click  Floating Vertical Curve (Through Point, Parameter).

3   Select the fixed or floating entity to which you want to attach the curve.

**4** Specify a pass-through point.

**5** If selected a parabola in step 3, specify whether the resulting curve should be either a crest or sag.

**6** Do one of the following:

- To specify a K value, enter the K value.

- To specify a minimum radius for the curve parabola, enter R (radius), and then specify the radius.

> **NOTE** If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at given design speeds are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

Floating Vertical Curve (Through Point, Parameter)

Dialog Box

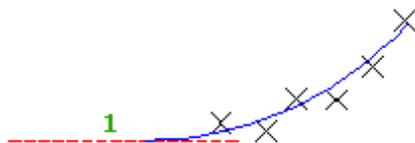Profile Layout Tools (page 2170)

## Adding a Floating Vertical Curve by Best Fit

Add a floating, three-point curve entity by best fit to a profile. You can define the best fit entity using a series of AutoCAD Civil points, existing entities, or clicks on screen.

The illustration shows a floating vertical curve created from entity 1 by best fit. The Xs indicate data points used to create the entity.

**NOTE** Each best fit profile entity is drawn to the vertical scale set in the profile view style. To create an accurate best fit entity, set the profile view style Vertical Exaggeration value to 1.000 to match the horizontal scale of the profile view. For more information, see Creating and Editing Profile View Styles (page 1030).

**See also:**

■ Editing Best Fit Profile Entities (page 1100)

**To add a floating vertical curve by best fit from AutoCAD Civil points**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

3 In the Profile Layout Tools (page 2170) toolbar, click Floating Vertical Curve - Best Fit.

4 Select an existing fixed or floating profile entity to which to attach the best fit tangent.

   If you select the first half of the profile entity, the best fit entity will precede the profile entity. If you select the second half of the profile entity, the best fit entity will succeed the profile entity.

5 In the Curve by Best Fit dialog box (page 1989), select From COGO Points.

6 In plan view, select two or more points. Enter G to select a point group or N to enter points by number.

   As the points are selected in plan view, an X marks each regression point and a temporary, dashed curve is displayed in the profile view.

7 Press Enter to complete the command.

8 In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

   As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

9 Create the curve:

   ■ Click to create the curve and keep the Regression Data vista open.

   ■ Click to create the curve and close the Regression Data vista.

**To add a floating vertical curve by best fit by clicking on screen**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

3 In the Profile Layout Tools (page 2170) toolbar, click Floating Vertical Curve - Best Fit.

4 Select an existing fixed or floating profile entity to which to attach the best fit tangent.

   If you select the first half of the profile entity, the best fit entity will precede the profile entity. If you select the second half of the profile entity, the best fit entity will succeed the profile entity.

5 In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen.

6 Select at least two points in the drawing.

   **NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**7** Press Enter to complete the command.

**8** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**9** Create the curve:

- ■ Click ![icon] to create the curve and keep the Regression Data vista open.

- ■ Click ![icon] to create the curve and close the Regression Data vista.

**To add a floating vertical curve by best fit from existing entities**

**1** Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

**2** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon] .

**3** In the Profile Layout Tools (page 2170) toolbar, click ![icon] Floating Vertical Curve - Best Fit.

**4** Select an existing fixed or floating profile entity to which to attach the best fit tangent.

If you select the first half of the profile entity, the best fit entity will precede the profile entity. If you select the second half of the profile entity, the best fit entity will succeed the profile entity.

**5** In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen. Specify the tessellation and mid-ordinate tolerance settings.

**6** Select one of the entities listed on the command line.

**NOTE** You can select several types of entities listed on the command line.

**7** If you selected a profile entity, specify the starting and ending station on the Specify Station Range dialog box (page 1992).

**8** Repeat steps 6 and 7 to select other entities from which to create the best fit curve.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**9** Press Enter to complete the command.

**10** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**11** Create the curve:

- ■ Click ![icon] to create the curve and keep the Regression Data vista open.

- ■ Click ![icon] to create the curve and close the Regression Data vista.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✏️

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

🔧 Floating Vertical Curve - Best Fit

Dialog Box

Profile Layout Tools (page 2170)
Curve by Best Fit (page 1989)
Regression Data (page 1990)

## Adding a Floating Vertical Curve with a Pass-Through Point and Grade

Add a floating vertical curve by specifying a pass-through point and a grade.

The illustration shows a floating curve from an existing entity to a pass-through point (2) with a grade (3).



**To add a floating vertical curve with a pass-through point and grade**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✏️ .

2 On the Profile Layout Tools toolbar, click 🔧 Floating Vertical Curve (Through Point, Grade).

3 Select the fixed or floating entity to which you want to attach the curve.

4 Specify a pass-through point.

5 Specify a grade.

**Quick Reference**

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

Floating Vertical Curve (Through Point, Grade)

Dialog Box

Profile Layout Tools (page 2170)

## Adding Free Vertical Curves

Add free vertical curves to build constraint-based profile geometry.

Free entities are defined by attachment to two entities, on which the free entity is dependent to define its geometry.

## Adding a Free Vertical Curve with a Parameter

Add a free vertical curve between two entities by specifying a parameter.

Specify a K value, length, or radius to define the curve.

The illustration shows a free curve between entities 1 and 2, specified with a length parameter 3.
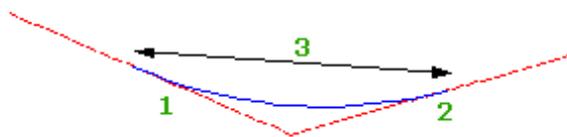


**To add a free vertical curve with a parameter**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   On the Profile Layout Tools toolbar, click  Free Vertical Curve (Parameter).

3   Select the first fixed or floating entity to which you want to attach the curve.

4   Select the second fixed or floating entity to which you want to attach the curve.

**5** Enter one of the following parameters to complete the curve definition:

- Length: Enter L

- K Value: Enter K

- Radius: Enter R

---

**NOTE** If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at given design speeds are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

---

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Free Vertical Curve (Parameter)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Free Parabolic Vertical Curve

Add a free parabolic vertical curve.

Specify the curve by length, a pass-through point, or a K value.

The illustration shows a free curve between entities 1 and 2, specified with a length parameter 3.
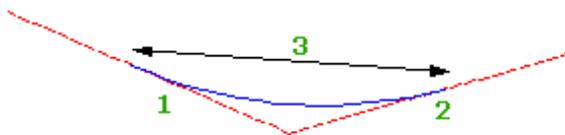


**To add a free parabolic vertical curve**

**1** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

**2** On the Profile Layout Tools toolbar, click ⊟ Free Vertical Parabola (PVI Based).

**3** In the profile view, select a location near the PVI to which you want to add a curve.

**4** Enter one of the following parameters to complete the curve definition:

- Length: Enter L.

- Pass-Through Point: Enter P

- K Value: Enter K.

**NOTE** If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at a given design speed are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ✏️

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

⊟ Free Vertical Parabola (PVI Based)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Free Asymmetrical Vertical Curve

Add a free asymmetrical vertical curve.

You specify the length of the curve before the PVI and the length of the curve after the PVI.

The illustration shows an asymmetrical curve between tangents 1 and 2, with curve lengths 3 and 4.
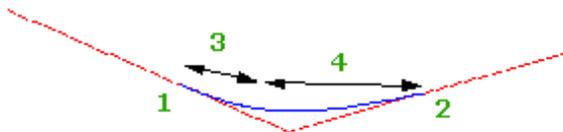
**To add a free asymmetrical vertical curve**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon].

2   On the Profile Layout Tools toolbar, click ![icon] Free Asymmetrical Parabola (PVI Based).

3   In the profile view, select a location near the PVI to which you want to add a curve.

4   Specify the length of the curve before the PVI by entering a value or by selecting two points in the drawing window.

    If the first length you specify is too large for the asymmetric curve, you cannot enter the length for the second curve. The length of the first curve must be less than the station distance between the PVI you select and the station value of the previous PVI (or end station of the previous curve.)

5   Specify the length of the curve after the PVI by entering a value or by selecting two points in the drawing window.

    **NOTE**  If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at given design speeds are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

## Quick Reference

Ribbon

  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon]

Menu

  Click Profiles menu ➤ Edit Profile Geometry

Command Line

  EditProfileLayout

Profile Layout Tools Toolbar Icon

  ![icon] Free Asymmetrical Parabola (PVI Based)

Dialog Box

  Profile Layout Tools (page 2170)

## Adding a Free Circular Vertical Curve

Add a free circular curve.

Specify a radius, length, or pass-through point to define the curve.

The illustration shows a circular curve at PVI 1, passing through point 2.
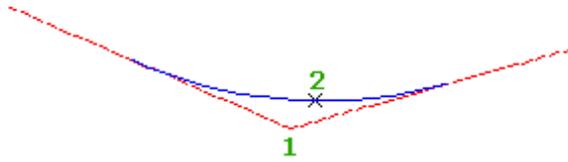
**To add a free circular vertical curve**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon] .

2   On the Profile Layout Tools toolbar, click ![icon] Free Circular Curve (PVI Based).

3   In the profile view, select a location near the PVI to which you want to add a curve.

4   Enter one of the following parameters to complete the curve definition:

■   Radius: Enter R.

■   Length: Enter L.

■   Pass-Through Point: Enter P.

> **NOTE** If the profile was created using the criteria-based design (page 2471) feature, a default value is displayed on the command line. Minimum K values at given design speeds are specified in the design criteria file (page 2472). Minimum curve length and radius are calculated from the minimum K value at the current design speed.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon]

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

![icon] Free Circular Curve (PVI Based)

Dialog Box

Profile Layout Tools (page 2170)

## Adding a Free Vertical Curve by Best Fit

Add a free, three-point curve entity by best fit to a profile. You can define the best fit entity using a series of AutoCAD Civil points, existing entities, or clicks on screen.

The illustration shows a free vertical curve created by best fit between endpoints 1 and 2. The Xs indicates data points used to create the entity.
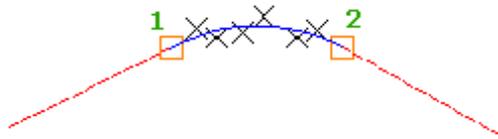


**NOTE** Each best fit profile entity is drawn to the vertical scale set in the profile view style. To create an accurate best fit entity, set the profile view style Vertical Exaggeration value to 1.000 to match the horizontal scale of the profile view. For more information, see Creating and Editing Profile View Styles (page 1030).

**See also:**

- Editing Best Fit Profile Entities (page 1100)

**To add a free vertical curve by best fit from AutoCAD Civil points**

1  Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 🖉 .

3  In the Profile Layout Tools (page 2170) toolbar, click ⛰ Free Vertical Curve - Best Fit.

4  Select an existing fixed or floating profile entity (the First Entity) from which to attach the best fit tangent.

5  Select an existing fixed or floating profile entity (the Next Entity) to which to attach the best fit tangent.

6  In the Curve by Best Fit dialog box (page 1989), select From COGO Points.

7  In plan view, select one or more points. Enter G to select a point group or N to enter points by number.

   As the points are selected in plan view, an X marks each regression point and a temporary, dashed curve is displayed in the profile view.

8  Press Enter to complete the command.

9  In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

   As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

10  Create the curve:

   - Click 💾 to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To add a free vertical curve by best fit by clicking on screen**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 📝 .

3 In the Profile Layout Tools (page 2170) toolbar, click 🔺 Free Vertical Curve - Best Fit.

4 Select an existing fixed or floating profile entity (the First Entity) from which to attach the best fit curve.

5 Select an existing fixed or floating profile entity (the Next Entity) to which to attach the best fit curve.

6 In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen.

7 Select one or more points in the drawing.

**NOTE** You can use OSNAP or transparent commands to select points.

As you select points in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

8 Press Enter to complete the command.

9 In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

10 Create the curve:

■ Click 💾 to create the curve and keep the Regression Data vista open.

■ Click ✔ to create the curve and close the Regression Data vista.

**To add a free vertical curve by best fit from existing entities**

1 Set the profile view style Vertical Exaggeration value to 1.0000. See Creating and Editing Profile View Styles (page 1030) for more information.

2 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 📝 .

3 In the Profile Layout Tools (page 2170) toolbar, click 🔺 Free Vertical Curve - Best Fit.

4 Select an existing fixed or floating profile entity (the First Entity) from which to attach the best fit tangent.

5 Select an existing fixed or floating profile entity (the Next Entity) to which to attach the best fit tangent.

6 In the Curve by Best Fit dialog box (page 1989), select By Clicking On The Screen. Specify the tessellation and mid-ordinate tolerance settings.

7 Select one of the entities listed on the command line.

**NOTE** You can select several types of entities listed on the command line.

**8** If you selected a profile entity, specify the starting and ending station on the Specify Station Range dialog box (page 1992).

**9** Repeat steps 7 and 8 to select other entities from which to create the best fit curve.

As you select entities in the drawing window, an X marks each regression point and a temporary, dashed curve is displayed in real time.

**10** Press Enter to complete the command.

**11** In the Panorama window, use the Regression Data vista (page 1990) to modify the regression points.

As you highlight a row in the Regression Data vista, the corresponding regression point in the drawing window is highlighted in red.

**12** Create the curve:

■ Click ![save icon] to create the curve and keep the Regression Data vista open.

■ Click ![check icon] to create the curve and close the Regression Data vista.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon]

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

![icon] Free Vertical Curve - Best Fit

Dialog Box

Profile Layout Tools (page 2170)
Curve by Best Fit (page 1989)
Regression Data (page 1990)

# Editing Layout Profiles

Use the profile layout tools to edit curves, tangents, and points of vertical intersection (PVIs) on layout profiles.

You can edit a layout profile by manipulating grips or by entering specific values that reflect road design criteria.

Selecting either the ![icon] PVI Based or ![icon] Entity Based button on the Profile Layout Tools (page 2170) toolbar has three effects:

■ Updates the tools available on the Profile Layout Tools toolbar

■ Changes the data shown in the Profile Entities vista (page 2166)

■ Changes the data shown in the Profile Layout Parameters Dialog Box (page 2168)

For information on profile tools for creating and editing profiles, specifying curve settings, and displaying the profile editors, see Profile Layout Tools (page 2170).

**TIP** You can use transparent commands when you are prompted to specify direction, radius, or length, or use station offsets to specify any points. For more information, see Transparent Commands (page 1581).

# Selecting Profile Sub-entities

Select an entire profile or a range of sub-entities in order to precisely edit them.

Select an entire profile by clicking it. Select an individual profile sub-entity or range or sub-entities by Ctrl+clicking.

After you have selected a profile entity, you can use a variety of editing tools.

If the Profile Entities vista (page 2166) and Profile Layout Tools (page 2170) toolbar are open, selecting a sub-entity or range of sub-entities in a profile causes the selected sub-entities to display in the Profile Entities vista. Selecting sub-entities or ranges of sub-entities in a different profile causes that range of selected sub-entities to display in the Profile Entities vista and have focus of the Profile Layout Tools toolbar. The last object selected becomes the current object and has focus of the profile editing tools.

**To select an entire profile**

1 Click the profile.

Grips are displayed along the profile.

2 Use the desired tools to edit the profile:

| To Edit the profile by... | see this topic: |
| --- | --- |
| adding constraint-based fixed, free, or floating tangents and curves | Adding Tangents and Curves to a Layout Profile (page 1059) |
| changing the profile at the points of vertical intersection | Editing PVIs (page 1093) |
| deleting profile sub-entities | Deleting Profile Sub-entities (page 1097) |
| using grips or entering precise parameter values | Editing Profile Curve and Tangent Parameter Values (page 1098) |

**To select a single profile sub-entity**

1 In the drawing window, click the profile.

2 Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ![icon] .

3 In the Profile Layout Tools toolbar (page 2170), click ![icon] Entity Based to enable the profile entity editing tools.

4 In the Profile Layout Tools toolbar, click ![icon] Profile Grid View.

The Profile Entities vista (page 2166) displays attributes for all profile sub-entities.

5 In the drawing window, Ctrl+click the desired profile sub-entity.

The selected sub-entity is highlighted in the drawing, and the Profile Entities vista displays attributes for only the selected profile sub-entity.

**6** Edit the profile sub-entity by entering specific attribute values in the Profile Entities vista.

See Editing Profile Curve and Tangent Parameter Values (page 1098) for more information.

**NOTE** The Profile Entities vista (page 2166) displays the attributes of only the selected sub-entity.

**To select a range of profile sub-entities**

**1** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ▧ .

**2** In the Profile Layout Tools toolbar (page 2170), click ⟋ Entity Based to enable the profile entity editing tools.

**3** In the Profile Layout Tools toolbar, click ▦ Profile Grid View.

The Profile Entities vista (page 2166) displays attributes for all profile sub-entities.

**4** In the drawing window, Ctrl+click the profile sub-entity at the beginning of the desired range.

The selected sub-entity is highlighted in the drawing, and the Profile Entities vista displays attributes for only the selected profile sub-entity.

**5** In the drawing window, Ctrl+click the profile sub-entity at the end of the desired range.

The selected range of sub-entities is highlighted in the drawing, and the Profile Entities vista displays attributes for only the selected profile sub-entities.

**6** Edit the profile sub-entities by entering specific attribute values in the Profile Entities vista.

See Editing Profile Curve and Tangent Parameter Values (page 1098) for more information.

**NOTE** The Profile Entities vista (page 2166) displays the attributes of only the selected sub-entities.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ▧

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

▦ Profile Grid View

Dialog Box

Profile Layout Tools (page 2170)

# Editing PVIs

Manage the elevations and grades along a layout profile or static surface profile by editing the points of vertical intersection (PVIs).

Adjusting the number and location of PVIs allows you to make major changes to a layout profile, such as removing vertical curves and changing tangent grades.

You can lock a PVI at a specific station and elevation so that it cannot be moved by edits to adjacent entities. This is often done in intersections, where a side road profile is locked to the crown and edge-of-pavement elevations of the main road. If you unlock a dynamic PVI for one road in an intersection, you break the link to the profile of the other road.

You can also move PVIs with editing grips. For more information about the use of editing grips, see Editing Profile Curve and Tangent Parameter Values (page 1098).

**To insert a PVI**

1   Click the profile. Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   In Profile Layout Tools (page 2170), click  Insert PVI.

3   Specify a point along the profile for the location of the new PVI. Specify the point either by clicking the point or by entering **'pse** and providing a numeric value.

4   Optionally, add other PVIs by repeating Step 4.

5   Right-click to finish.

**To insert multiple PVIs (bulk data entry)**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   In Profile Layout Tools (page 2170), click  Insert PVIs - Tabular.

3   To place a curve at each inserted PVI, click the desired curve type in the Insert PVIs dialog box (page 2164). Otherwise, click None. The choice made here applies to all PVIs inserted at this time.

4   Click the first Station cell to activate it. Enter station and elevation values for each PVI. Press Enter after each value to move the cursor to the next field. To change a value after entering it in the table, click the cell and modify the entry.

5   After all values are entered, check the table to ensure that each value is correct.

6   Click OK to insert the set of PVIs.

**To delete a PVI**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   In Profile Layout Tools dialog box (page 2170), click  Delete PVI.

3   In the profile view, click near the PVI you want to delete. The PVI is removed, and the tangents are adjusted to connect the adjacent PVIs or endpoints.

**To lock a PVI**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2   In theProfile Layout Tools dialog box (page 2170), select the PVI based display option.

3   Click  Profile Grid View.

4   Find the PVI to lock, and optionally edit its location by changing numeric data in the table.

**5** Click the open lock icon in the table row for the PVI. The icon changes to a closed lock state, and some values in the table are dimmed to indicate they cannot be edited.

**To unlock a PVI**

**1** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**2** In the Profile Layout Tools dialog box (page 2170), select the PVI based display option.

**3** Click  Profile Grid View.

**4** Click the closed lock icon in the table row for the PVI. The icon changes to an open lock state, and additional values in the table can be edited.

**TIP** You can also unlock a PVI by clicking the lock icon in the profile view.

**To move a PVI**

**1** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**2** In Profile Layout Tools (page 2170), click  Move PVI.

**3** In the drawing window, click near the PVI that you want to move.

**4** Click the new location for the PVI. The PVI is immediately moved to the new location. The adjacent tangents are adjusted accordingly.

**5** To move additional PVIs, repeat steps 4 and 5.

**To raise or lower a series of PVIs**

**1** Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

**2** In Profile Layout Tools (page 2170), click  Raise/Lower PVIs.

**3** In the "Raise/Lower PVI Elevations dialog box", enter the elevation distance to raise or lower the PVIs. A positive number raises the PVIs and a negative number lowers them.

**4** Click the PVI Range you want to move: either All PVIs or those within a range of stations.

**5** If you clicked Station Range, specify the start and end stations.

**6** Click OK to move the PVIs.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles menu ➤ Edit Profile Geometry

Object Shortcut Menu

Right-click <profile object> ➤ Edit Profile Geometry

EditProfileLayout
AddProfilePVI
DeleteProfilePVI

Dialog Box

Profile Layout Tools (page 2170)

# Converting AutoCAD Entities to Profile Sub-entities

Convert an AutoCAD line or spline to a profile sub-entity.

AutoCAD lines and curves can be created using the best fit entities or similar commands. See Lines and Curves (page 559) for more information.

The converted sub-entity becomes part of the solved profile geometry if either of the following two conditions are met:

■ Before it is converted, it must be attached to an unattached end point of another solved entity in the profile.

■ After it has been converted, it may be joined to the solved profile geometry using the profile layout tools.

**To convert an AutoCAD entity to a profile sub-entity**

1   Create a profile view or select an existing profile view.

2   Set the profile view style vertical exaggeration to 1. For more information, see Creating and Editing Profile View Styles (page 1030).

3   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

4   On the Profile Layout Tools (page 2170) toolbar, click Convert AutoCAD Line And Spline.

5   Select the AutoCAD entity.

**NOTE** If the sub-entity you converted is hidden behind the source AutoCAD entity, use Shift+spacebar to select it. Hover the cursor over the white AutoCAD entity. Press and hold Shift, then press the spacebar to highlight the profile sub-entity. Click to select the profile sub-entity, then release the Shift key. Click the profile sub-entity again to select it and continue with the command.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

Convert AutoCAD Line And Spline

# Deleting Profile Sub-entities

Use the Profile Layout Tools dialog box (page 2170) to remove tangents and curves from a layout profile.

When you delete a curve, the tangents on either side of the curve are preserved. If the tangents were joined when they were created, the tangents are joined at the PVI after the curve is deleted.

---

**NOTE** You cannot delete a tangent that has a curve attached to it. **Before deleting a tangent,** you must first delete the PVI and any dependent curves and tangents.

---

You can delete profile sub-entities either using the Profile Layout Tools dialog box (page 2170) or selecting the sub-entity and pressing Delete.

**To delete a sub-entity from the Profile Layout Tools dialog box**

1  Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor  .

2  In the Profile Layout Tools dialog box (page 2170), click  Delete Entity.
   If you do not see the button, click

    Entity Based to change to an entity-based view.

3  In the drawing window, select a sub-entity to delete.

**To delete a sub-entity using the Delete key**

1  Ctrl+click the profile sub-entity you want to delete.

2  Press Delete.

## Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Object Shortcut Menu

Right-click <profile object>  ➤  Edit Profile Geometry

Command Line

EditProfileLayout

Dialog Box

Profile Layout Tools (page 2170)

Profile Layout Tools Dialog Box Icon

 Delete Entity

# Editing Profile Curve and Tangent Parameter Values

Use the Profile Entities vista and Profile Layout Parameters dialog box to display and edit parameters of profile entities that are part of the solved profile geometry.

The Profile Entities vista displays a form that is like a spread sheet, where each row represents either a profile entity or PVI and each column represents a parameter. Use the △ PVI Based and △ Entity Based buttons on the Profile Layout Tools toolbar to change the type of data that is displayed on the Profile Entities vista. In the Profile Entities vista, you can right-click a column heading to display options for customizing the display of detail in the column views.

The purpose of Profile Entities vista is to let you quickly scroll through the entities that make up the profile. The list displays only the entities that have been selected (page 1092) in the drawing window. When you select a row, the details for the corresponding entity are displayed in the Profile Layout Parameters dialog box.

The Profile Layout Parameters dialog box enables you to view and change the many attributes for a single profile sub-entity. The Profile Layout Parameters dialog box contains up to three panels. The Layout Parameters panel, which displays the geometric parameters of the selected sub-entity, is always present. In this panel, the Value column lists the actual value of each sub-entity Parameter. If the profile was created using the criteria-based design (page 2471) feature, the Constraint column lists the value to which a Parameter is constrained by the specified design criteria. Use the Value and Constraint columns together to view whether the sub-entity parameters meet the specified design criteria. If the profile was not created using the criteria-based design feature, the Constraint column is not available.

Two additional panels may be present on the Profile Layout Parameters dialog box if the profile has design criteria applied to it. If the profile has a design criteria file (page 2472) associated with it, the criteria from the file are displayed in the Design Criteria panel. If the profile has a design check set (page 2472) associated with it, the name and contents of the design check set are displayed in the Design Checks panel.

---

**NOTE** The attributes of a curve or other profile entity that you can edit depends on the method you used to create the entity. Generally speaking, you can edit the numeric value of a parameter if the parameter was used to define the entity. For all other parameters, numeric values are calculated and displayed but cannot be edited.

---

**See also:**

■ Vertical Curve Design (page 1068)

■ Editing Best Fit Profile Entities (page 1100)

**To edit profile layout parameters using the Profile Entities vista and Profile Layout Parameters dialog box**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬇ .

2   In the Profile Layout Tools (page 2170), click 📝 Profile Layout Parameters and 🔲 Profile Grid View.

3   Specify the type of data to display in the Profile Entities vista:

   ■ To display PVI data, click △ PVI Based.

   ■ To display entity data, click △ Entity Based.

   The Profile Entities vista (page 2166) displays the attributes of the selected profile entities. The Profile Layout Parameters dialog box (page 2168) is empty.

4   In the Profile Entities vista, select the specific PVI, entity, or sub-entity you want to analyze. The parameters of the selected the entities are displayed in the Profile Layout Parameters dialog box. Editable values are in black text (not shaded).

**5** To display a range of entities in the Profile Entities vista, Ctrl+click the entity at the end of the desired range.

The selected range of entities is displayed on the Profile Entities vista. Highlighting a row displays the parameters of that entity in the Profile Layout Parameters dialog box.

**NOTE** Selecting another profile causes the Profile Layout Tools dialog box, Profile Entities vista, and Profile Layout Parameters dialog box to become active for the new profile. See Selecting Profile Sub-entities (page 1092) for more information about profile sub-entity selection.

**6** Review and edit the data as required. To edit another PVI, entity sub-entity, or range, repeat Steps 5 and 6.

**7** When finished, close the Profile Layout Parameters dialog box and Profile Layout Tools toolbar.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Dialog Boxes

Profile Layout Tools (page 2170)
Profile Entities Vista (page 2166)
Profile Layout Parameters (page 2168)

Profile Layout Tools Dialog Box Icon

Profile Layout Parameters

Profile Grid View

PVI Based

Entity Based

## Editing Layout Profiles Using Grips

Use grips to graphically change the vertical curves and tangents in a profile.

Use editing grips to relocate or modify curves and tangents using visual design cues. If you created profile entities using the constraint-based tools, some editing grips are not available. For example, if you attach a floating curve to a fixed or floating tangent, the editing grip is not available for editing the PVI of the two entities.

**NOTE** When a grip is selected, you can use either dynamic input or the Profile Entities (page 2166) and Profile Layout Parameters (page 2168) dialog boxes to enter precise attribute values for the grip.

**To grip edit layout profiles**

1  Click a profile object in a profile view. If the profile type is static, editing grips appear on the tangents and vertical curves. Grip behavior depends on the type of grip:

| Grip Type | Behavior |
|---|---|
| ▲ at point of vertical intersection (PVI) | Moves the PVI, changing grades and lengths of adjacent tangents.<br>Preserves curve length and radius.<br>Use with ORTHO mode turned on to move PVI horizontally or vertically. |
| ▶ at end of tangent | Adjusts the tangent length while preserving the grade.<br>Modifies the grade of adjacent tangent.<br>Preserves curve length and radius. |
| ■ at end of tangent | Adjusts the tangent length and grade simultaneously.<br>Modifies the grade of adjacent tangent. |
| ■ at tangent mid-point | Moves the entire tangent, without changing the length or grade.<br>Changes the location of adjacent PVIs and the grade of adjacent tangents.<br>Preserves the radius and length of adjacent curves. |
| ● at curve endpoint | Moves the endpoint along the tangent, without affecting the tangent grades or PVI location.<br>Changes position of both curve endpoints so that midpoint remains near PVI.<br>Changes the curve length and radius. |
| ● at curve midpoint | Moves a pass-through point for the curve, changing the curve length, radius, and position of both endpoints as required.<br>PVI location is not affected.<br>After the curve is edited, this grip moves to the new midpoint. |

2  Identify the grip you want to move, then click it once. The grip changes color from cyan to red.

3  Drag the grip to its new location, and click to place it there. The grip follows your mouse pointer, within the constraints imposed by the grip type.

**NOTE** When a grip is selected, you can use either dynamic input or the Profile Entities vista (page 2166) and Profile Layout Parameters dialog box (page 2168) to enter precise attribute values for the grip.

# Editing Best Fit Profile Entities

Use the Regression Data and Profile Entities vistas to edit parameters of profile entities that were created by best fit.

The Regression Data vista displays the source regression data from which the best fit entity was created. The regression data can be modified either before or after the entity has been created. The information in the Regression Data vista can be changed, and then applied to the entity. If the entity parameters are changed outside the Regression Data vista, the entity can be synchronized to the current regression data.

The Profile Entities vista displays a form that is like a spread sheet, where each row represents a profile entity and each column represents a parameter. When an entity is created by best fit, Best Fit is displayed in the Constraint2 column. Click [...] in this column to edit the source regression data.

Regression data is not dynamically linked to the best fit entity. If the entity is edited, ▽ appears in both the Regression Data and Profile Entities vistas.

**NOTE** You can edit only values that are available, you cannot edit unavailable (shaded) text.

**Editing Floating Best Fit Entities**

The manner in which a best fit floating entity reacts to edits to the attachment entity depends on whether the best fit entity is synchronized with the best fit sample data.

■ If the floating best fit entity is synchronized with the best fit sample data, it maintains tangency when the attachment entity is edited. The floating best fit entity geometry changes to adapt to changes to the attachment entity, and continue to honor the regression points.

■ If the floating best fit entity is not synchronized with the best fit sample data, it behaves as a normal floating entity. The floating best fit entity remains tangent to the attachment entity, and its geometry will not change to honor the regression points.

**To edit best fit regression data**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

2 On the Profile Layout Tools toolbar, click Edit Best Fit Data For All Entities.

**NOTE** You can also open the Regression Data vista from the Profile Entities vista. In the Constraint2 column, click [...] .

3 In the Regression Data vista, edit the regression data as needed.
You can add, edit, or remove regression points.

4 Click ✔ to apply the changes and close the Regression Data vista.

**To synchronize a best fit profile entity to the regression data**

1 Select the profile that contains the best fit entity.

2 Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor .

3 On the Profile Layout Tools toolbar, click Edit Best Fit Data For All Entities.

4 In the Regression Data vista, click ▽ Synchronize Entity To Best Fit Sample Data.

**NOTE** You can also synchronize the entity to the regression data from the Profile Entities vista. In the Constraint2 column, click ▽ .

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 

Menu

Click Profiles  menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Toolbar Icon

 Edit Best Fit Data For All Entities

Regression Data Vista Icon

 Synchronize Entity To Best Fit Sample Data

Profile Entities Vista Icon



Dialog Box

## Applying Design Criteria to an Existing Layout Profile

Use the Profile Properties dialog box to apply design criteria to an existing profile.

The criteria-based design feature provides the ability to verify that your profile design meets the minimum standards required by your local agency.

For more information, see Criteria-Based Profile Design (page 1044).

**To apply design criteria to an existing profile**

1   Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Profile Properties  .

2   In the Profile Properties dialog box, click the Design Criteria tab (page 2175).

   **NOTE** To apply design criteria to a profile, there must be at least one design speed applied to the profile from which the profile was created.

3   Select the Use Criteria-Based Design check box.

4   Specify the following settings:
   ■   Use Design Criteria File
       Select this check box to associate a design criteria file with the profile. The design criteria file field and the Default Criteria properties become available when you select the check box.

       **NOTE** The design criteria file that is applied to the parent alignment is used by default. A separate design criteria file can be applied to the profile.

■ Use Design Check Set
Select this check box to associate a design check set with the profile. The design check set list becomes available when you select the check box. Select a design check set from the list.

**5** Click OK.

> **NOTE** The entire profile design is validated when you click either Apply or OK.

**Quick Reference**

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Profile Properties drop-down ➤ Profile Properties

Toolspace Shortcut Menu

Prospector tab: Right-click <profile item > ➤ Properties

Object Shortcut Menu

Right-click <profile object> ➤ Profile Properties

Command Line

EditProfileProperties

Dialog Box

Profile Properties (page 2174).

# Viewing and Correcting Profile Design Criteria Violations

Use the warning symbols to verify whether the profile design meets the specified criteria and design checks.

When a profile uses the criteria-based design feature, the profile geometry must meet the design criteria specified in the design criteria file (page 2472) and design checks (page 2472).

When a profile sub-entity violates the design criteria, a warning symbol is displayed in the following locations:

■ The drawing window — The ⚠ warning symbol is displayed on each sub-entity that violates the design criteria.

> **NOTE** The size of the warning symbol does not adjust automatically. After zooming in or out, enter REGEN to resize the warning symbols.

■ The Profile Entities vista — The ⚠ warning symbol is displayed in the No. column of the sub-entity that violates the design criteria. If a criteria in the design criteria file was violated, the warning symbol is also displayed next to the specific criteria and parameter value that violates the design criteria.

■ The Profile Layout Parameters dialog box — If a criteria in the design criteria file was violated, the ⚠ warning symbol is displayed in the Design Criteria panel next to the Property that was violated, and in the Layout Parameters panel next to the Parameter that was violated. If a design check was violated, the ⚠ warning symbol is displayed in the Design Checks panel next to the design check that was violated.

You can use the ⚠ warning symbols in the drawing window to view information about criteria and design checks that were violated. When the cursor is hovered over a warning symbol in the drawing window, a tooltip displays information about the violation. If a design criteria was violated, the tooltip displays the criteria that has been violated, as well as the minimum value required to meet the criteria. If a design check was violated, the tooltip displays the name of the design check that has been violated.

**BEST PRACTICE** The name of the design check should be similar to the formula you enter in the Design Check field. When a design check is violated, only the design check name is displayed, and not the minimum acceptable value. If the formula is in the design check name, it is easier to correct the violation.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

## Turning Off the Profile Warning Symbol

Change the visibility of the warning symbol that indicates violations in the profile design.

There are several ways to turn off the warning symbol. The following methods are listed in the order in which they are recommended.

### Modify the Profile Geometry

Edit the profile design to meet the specified design criteria. This option clears the warning symbols from the drawing window, Profile Entities vista, and Profile Layout Parameters dialog box.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

For more information, see Correcting Profile Design Criteria Violations (page 1105).

### Change the Profile Style

Turn off the Warning Symbol display component in the current profile style. Alternatively, create a separate profile style that has the Warning Symbol display component turned off. This option clears the warning symbols from the drawing window for profiles that use that style. The warning symbols are still displayed in the Profile Entities vista and Profile Layout Parameters dialog box.

For more information, see Profile Styles (page 1027).

### Turn Off the AutoCAD Civil Solution Tip

The warning symbol is controlled by the Solution Tip setting in the Options dialog box on the AEC Editor tab. To hide the warning symbol for all drawings, clear the Drafting check box. This option clears all alignment and profile warning symbols from the drawing window. However, the warning symbols are still displayed in the Profile Entities vista and Profile Layout Parameters dialog box.

For more information, see AEC Editor Tab (Options Dialog Box) (page 2453).

**WARNING** Settings in the Options dialog box affect the overall behavior of AutoCAD Civil. If you clear the Solution Tip Drafting check box, the warning symbol for all alignment and profile objects is hidden in every drawing that is opened on the current installation of AutoCAD Civil.

**Turn Off the Criteria-Based Design Feature**

Disassociate the design criteria from the profile. The profile design check set and/or the design criteria file can be turned off in the Profile Properties dialog box. This option clears the warning symbols from the drawing window, Profile Entities vista, and Profile Layout Parameters dialog box.

For more information, see Applying Design Criteria to an Existing Layout Profile (page 1102).

**WARNING** If you turn off the criteria-based design feature, the profile design will not be validated against any design standards.

# Correcting Profile Design Criteria Violations

Validate the profile geometry against the specified criteria and design checks.

When a profile uses the criteria-based design feature, the profile geometry must meet the design criteria specified in the design criteria file (page 2472) and design checks (page 2472).

When a profile sub-entity violates the design criteria, a warning symbol is displayed in several locations. For information about where the warning symbols appear, see Viewing and Correcting Profile Design Criteria Violations (page 1103).

You can use the ⚠ warning symbols in the drawing window to view information about criteria and design checks that have been violated. When the cursor is hovered over a warning symbol in the drawing window, a tooltip displays information about the violation.

■ If a design criteria was violated, the tooltip displays the criteria that was violated, as well as the minimum value required to meet the criteria.

■ If a design check was violated, the tooltip displays the name and formula of the design check that was violated.

**NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed in the drawing. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria and design checks.

**To correct profile design criteria violations using grips**

1 In the drawing window, hover the cursor over a ⚠ warning symbol.

   A tooltip displays information about the violation.

2 Select the profile.

3 Use the profile grips to modify the profile geometry.

   The ⚠ warning symbol disappears when the sub-entity meets the design criteria.

   **NOTE** If a sub-entity violates more than one criteria or design check, only one warning symbol is displayed on the sub-entity. To clear the warning symbol, the sub-entity must be modified to meet all the design criteria.

**To correct profile design criteria violations by entering numeric values**

1 Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor 📝 .

**2** On the Profile Layout Tools toolbar, click ⊞ Profile Grid View.

**3** In the Profile Entities vista, click the row that contains the ⚠ warning symbol in the No. column.

The sub-entity parameters are displayed in the Profile Layout Parameters dialog box.

**4** In the Profile Layout Parameters dialog box, locate the ⚠ warning symbol in either of the following panels:

■ **Layout Parameters panel** — In the Value column, enter a numeric value that meets the value specified in the Constraints column.

■ **Design Checks panel** — Notice the design check that has been violated. Modify the profile design to meet the parameters specified in the design check.

### Quick Reference

Ribbon

Click the profile. Click Profile tab ➤ Modify Profile panel ➤ Geometry Editor ⬐

Menu

Click Profiles menu ➤ Edit Profile Geometry

Command Line

EditProfileLayout

Profile Layout Tools Dialog Box Icon

⬐ Profile Layout Parameters

⊞ Profile Grid View

Dialog Boxes

# Working with Profile Views

Use the profile view grid to display one or more profiles for a horizontal alignment.

You can configure data bands and profile annotations in a profile view to make it clearer or more informative for the user.

A profile can be presented in either single, multiple, or stacked profile views:

■ A single profile view is typically used to design and edit a profile. It displays the specified station range of the corresponding alignment in a single profile view grid.
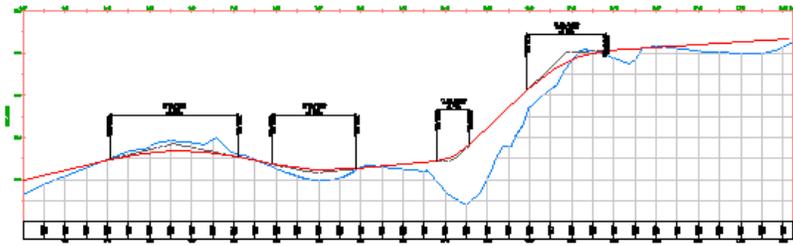
■ Multiple profile views are useful for plotting shorter segments of a profile in individual profile view grids of a consistent length and vertical scale.
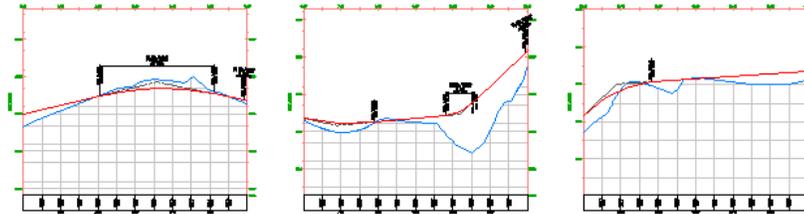


■ Stacked profile views are a collection of related profiles drawn in separate, vertically arranged profile views. Typically a centerline profile is contained in one profile view, and its left and right offsets are drawn in profile views that are placed above and below the centerline profile view.
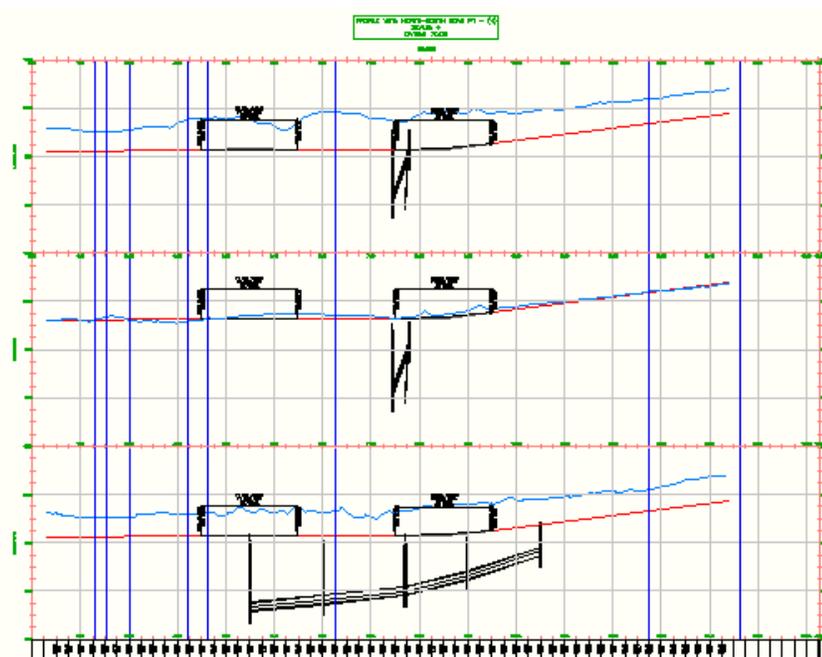


A profile can be split within either a single or multiple profile view. This allows a profile view to display a profile elevation range that is greater than the specified height of the profile view.
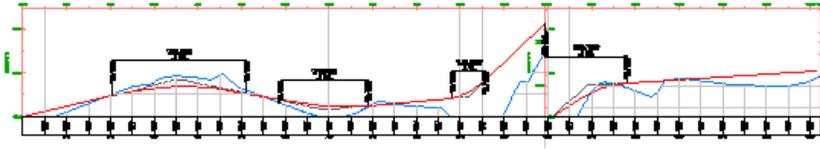
See The Profile View Object (page 1021) for more information.

# Creating Profile Views

Create a profile view to display a surface profile and offsets.

A profile view also provides the grid on which to draw a layout profile.

You can create profiles without displaying them on a profile view. When you create a profile view, you can display any of the profiles created from a selected alignment.

The default name format for profile views is PV - (1), PV - (2), etc. You can edit the name format to suit your business requirements.

An easy-to-use wizard guides you through the process of creating single, multiple, or stacked profile views.

Use multiple profile views to create final construction documents from a design. For best results, design your profile in a single profile view, and then create multiple profile views for plotting.

**See also:**

■ Plan Production Tools (page 1691)

## Creating a Single Profile View

Create a single profile view to display a surface profile and to design a layout profile.

Each profile view displays new or existing profiles and offsets for one horizontal alignment.
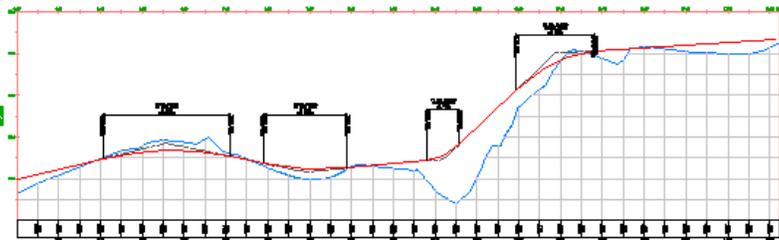


**See also:**

■ Creating Multiple Profile Views (page 1109)

■ Creating Stacked Profile Views (page 1111)

■ Creating a Superimposed Profile (page 1054)

**To create a single profile view**

1 Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Create Profile View .

**2** In the Create Profile View wizard (page 2145), navigate through the pages by using the links at the left or clicking Back or Next.

**NOTE** Click Create Profile View at any time to accept the current settings.

The wizard pages contain the following controls:

■ General page (page 2146) - Specify the alignment name and profile view name and description.

**NOTE** The Show Offset Profiles By Vertically Stacking Profile Views is only applicable when creating stacked profile views.

■ Station Range page (page 2146) - Specify the starting and ending stations.

■ Profile View Height page (page 2146) - Specify the profile view height and any split profile options, including split stations and styles for individual stations.

■ Profile Display Options page (page 2148) Specify the profiles to be drawn as well as their styles, labels, and layers.

■ Pipe Network Display page (page 2149) - Specify the pipe networks or parts to be drawn.

■ Data Bands page (page 2150) - Specify the data band sets and their properties.

■ Hatch Options page (page 2151) - Specify how to mark areas between two profiles on the profile view, such as where terrain must be cut or filled to create the design profile.

**3** Click Create Profile View.

**4** In the drawing window, click a location for the lower left corner (origin) of the profile view grid. The profile view is drawn.

## Quick Reference

Ribbon

Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Create Profile View 

Menu

Profiles menu ➤ Create Profile View

Command Line

CreateProfileView

Wizard

Create Profile View (page 2145)

## Creating Multiple Profile Views

Create multiple profile views to plot shorter segments of a profile in individual profile view grids of a consistent length and vertical scale.

Each profile view in the series is an independent profile view object. Each profile view object appears in Prospector, and you can modify the properties of each object independently.

**NOTE** Multiple profile views are most useful when you are creating final construction documents from your design. For best results, design your prof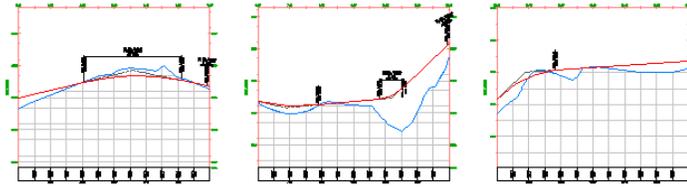ile in a single profile view, then use the Plan Production Tools (page 1691) to create multiple profile views for plotting.



**To create multiple profile views**

1 Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Create Multiple Profile Views 🖼️ .

2 In the Create Multiple Profile Views wizard (page 2145), navigate through the pages by using the links at the left or clicking Back or Next.

**NOTE** Click Create Profile Views at any time to accept the current settings.

The wizard pages contain the following controls:

■ General page (page 2146) - Specify the alignment name and profile view name and description.

**NOTE** The option Show Offset Profiles By Vertically Stacking Profile Views is applicable only when creating stacked profile views.

■ Station Range page (page 2146) - Specify the starting and ending stations.

■ Profile View Height page (page 2146) - Specify the profile view height and any split profile options, including split stations and styles for individual stations.

■ Profile Display Options page (page 2148) Specify the profiles to be drawn as well as their styles, labels, and layers.

■ Pipe Network Display page (page 2149) - Specify the pipe networks or parts to be drawn.

■ Data Bands page (page 2150) - Specify the data band sets and their properties.

■ Hatch Options page (page 2151) - Specify how to mark areas between two profiles on the profile view, such as where terrain must be cut or filled to create the design profile.

■ Multiple Plot Options page (page 2152) - Specify how the multiple profile views will be laid out.

3 Click Create Profile Views.

4 In the drawing window, click a location for the origin of the profile view grid. The profile view is drawn.

**NOTE** The profile view grid origin is determined by the Start Corner setting on the Multiple Plot Options page.

## Quick Reference

# Creating Stacked Profile Views

Create stacked profile views to display a centerline profile and related offset profiles in vertically arranged profile views.

Each profile view in the series or stack is an independent profile view object. Each profile view object appears in Prospector, and you can modify the properties of each object independently.
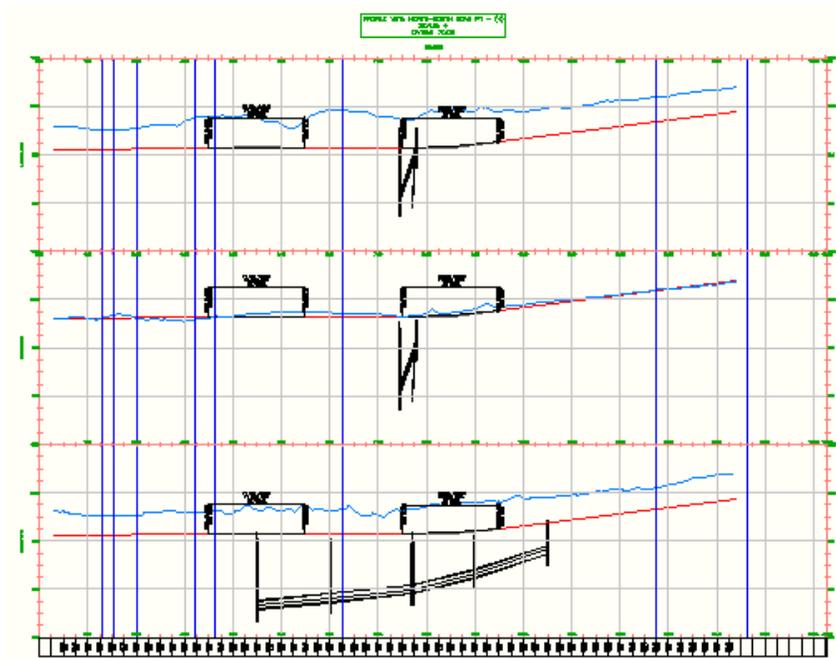


**To create stacked profile views**

1   Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Create Profile View ![icon]
    .

    OR

    Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Create Multiple Profile
    Views ![icon] .

**2** In the Create Profile View Wizard (page 2145) wizard, on the General page (page 2146), select the Show Offset Profiles By Vertically Stacking Profile Views check box.

The link to the Stacked Profile page (page 2147) is displayed on the left.

**3** On the General page, specify the alignment name and profile view name and description.

> **NOTE** Click Create Profile Views at any time to accept the current settings.

**4** On the Station Range page (page 2146), specify the starting and ending stations.

**5** On the Profile View Height page (page 2146), specify the profile view height and any split profile options, including split stations and styles for individual stations.

> **NOTE** If the profile elevation values are relatively consistent, under the Profile View Datum By option, select Mean Elevation. The Mean Elevation option vertically centers the profile line in the profile view, creating equal space above and below the profile line.

**6** On the Stacked Profile page, specify the number of stacked profile views, gap between the views, and the profile view style to use in the top, middle, and bottom views.

> **NOTE** The top and bottom profile views can use separate styles. All profile views between the top and bottom must be created with the same style.

**7** On the Profile Display Options page (page 2148), select a profile view from the Select Stacked View To Specify Options For list.

**8** In the Specify Profile Display Options table, specify the profiles to be drawn as well as their styles, labels, and layers.

**9** Repeat steps 7 and 8 for each profile view in the stack.

**10** On the Pipe Network Display page (page 2149), specify which pipe networks or parts will be drawn in each stacked profile view. Follow the workflow described in steps 7 through 9 to specify the options for each stacked profile view.

**11** On the Data Bands page (page 2150), specify the data band sets and their properties.

The data bands will appear at the top or bottom of the stack.

**12** For multiple, stacked profile views, on the Multiple Plot Options page (page 2152), specify how the multiple profile views will be laid out.

**13** Click Create Profile Views.

**14** In the drawing window, click a location for the origin of the stack of profile views.

> **NOTE** The profile view grid origin is determined by the Start Corner setting on the Multiple Plot Options page.

## Quick Reference

Menu

Profiles menu ➤ Create Profile View
Profiles menu ➤ Create Multiple Profile Views

Command Line

CreateProfileView

CreateMultipleProfileView

Wizard

Create Profile View (page 2145)

# Editing Profile Views

Edit profile views to change their contents, properties, and styles.

Use two dialog boxes to edit profile views:

■ Use the Profile View Properties dialog box (page 2178) to specify the name, profile view style, profile extents, included profile lines, data bands, and hatch options.

■ Use the Profile View Style dialog box (page 2185) to specify the annotation, vertical exaggeration, profile direction, grid format, and displayed elements.

You can change properties for an individual profile view. However, any changes in the Profile View Style dialog box affect all profile views that use that style. Therefore, if you must change one or more style items, create a style that can be applied where required.

Typical editing activities:

■ Add or delete profile lines.

■ Change the display style.

■ Add or delete data bands that display data such as stations, elevations, and horizontal geometry.

In the Profile View Properties dialog box, use the Draw check box to turn existing profiles on or off in the profile view. To delete a profile from the drawing, select it in the profile view and press the Delete key.

If you want to superimpose a profile from another alignment on a profile view, see Creating a Superimposed Profile (page 1054).

## Adding Profile Lines to a Profile View

Add profile lines to an existing profile view grid.

This procedure assumes that the profile lines you want to add have already been created in the drawing. To create a profile, see Creating Surface Profiles (page 1047) or Creating Layout Profiles (page 1048).

**To add profile lines to a profile view**

1 Click the profile view grid.

2 Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚ .

3 In the Profile View Properties Dialog Box (page 2178), click the Profiles tab (page 2179).

4 In the tables of profiles, select the Draw check box for each profile you want to add to the profile view.

5 Click OK.

**Quick Reference**

Ribbon

Click profile view. Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚

Object Shortcut Menu

Right-click profile view object ➤ Profile View Properties

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <Site name> ➤ Alignments ➤ <Alignment name> ➤ Profile Views, right-click <profile view name> ➤ Properties

Dialog Box

Profile View Properties (page 2178)

# Changing a Profile View Style

Change the appearance of a profile view by changing its style.

The profile view style controls the format of the graph on which the profiles are displayed, as well as the title and annotations on the axes.

Profile view styles are useful when your project transitions to a new phase. In the design phase, you can use one profile style that has few grid elements and limited annotation. Limiting annotation and grid elements reduces drawing size and improves drawing performance. A second style can be used for plotting. The plotted profile view style can be extensively annotated and use more graph elements than the design style.

If you are working with a split profile view, the Object Style field on the Profile View Properties - Information tab (page 2178) is ignored. Use the Profile View Style fields on the Elevations tab (page 2179) to change split profile view styles.

**See also:**

■ Profile View Styles (page 1029)

**To change a profile view style**

1  In the drawing window, select the profile view.

2  Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚.

3  In the Profile View Properties dialog box, on the Information tab (page 2178), select a style from the Object Style list.

4  Click OK.

**To change a split profile view style**

1  In the drawing window, select the profile view.

2  Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚.

3  In the Profile View Properties Dialog Box (page 2178), click the Elevations tab (page 2179).

**4** In the Split Profile View area, select a different style for the desired segment by name from the Profile View Style list.

> **NOTE** The start, intermediate, and end segments can each use a different style.

## Quick Reference

Ribbon

Click profile view. Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties 

Object Shortcut Menu

Right-click profile view object ➤ Profile View Properties

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <Site name> ➤ Alignments ➤ <Alignment name> ➤ Profile Views, right-click <profile view name> ➤ Properties

Command Line

EditGraphProperties

Dialog Box

Profile View Properties (page 2178)

## Adding Data Bands to a Profile View

Add data bands that display data such as stations, elevations, and horizontal geometry.

Place data bands along the top or bottom of a profile view grid to annotate the various profile lines with station and elevation data, vertical or horizontal geometry points, or superelevation, sectional, or pipe network data. You can also group a set of commonly used data bands into a band set that can be applied to a profile view as a single selection.

> **NOTE** The individual geometry points to label can be specified with the Profile Data band type.

**See also:**

■ Profile View Band Styles (page 1038)

■ Profile View Band Sets (page 1040)

■ Staggering Data Band Labels (page 1528)

**To add data bands to a profile view**

**1** Click the profile view grid.

**2** Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties .

**3** In the Profile View Properties Dialog Box (page 2178), click the Bands tab (page 2181).

**4** In the List Of Bands area, select either the bottom or the top of the profile view as the location for the new data band.

**5** Select a band type and style. Click Add.

If you add the Profile Data band type, use the Geometry Points to Label in Band dialog box (page 2162) to specify the individual alignment and profile points to label with the selected style.

**6** If you want to add more bands to the profile view, repeat steps 3 and 4. The List Of Bands is arranged in the same order as the actual bands on the profile view.

**7** Use the three buttons on the right side of the List Of Bands to arrange the bands as desired. You can move selected data bands up ⇧ or down ⇩ in the stack, and delete unwanted bands ✖ .

**8** Optionally, click Save As Band Set to save this arrangement of data bands as a set for reuse with other profile views. For more information, see Profile View Band Sets (page 1040).

**9** Click OK. The bands are displayed in the profile view.

### Quick Reference

Ribbon

Click profile view. Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties ⬚

Object Shortcut Menu

Right-click profile view object ➤ Profile View Properties

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <Site name> ➤ Alignments ➤ <Alignment name> ➤ Profile Views, right-click <profile view name> ➤ Properties

Command Line

EditGraphProperties

Dialog Box

Profile View Properties (page 2178)

## Adding Hatch Areas to a Profile View

Add hatch areas to highlight cut and fill regions along the profiles.

You can choose to hatch cut areas, fill areas, and areas with multiple surfaces as boundaries. You can also import quantity takeoff (QTO) criteria as a basis for defining hatch areas.

For each hatch area type, specify which profiles to use for upper and lower boundaries, and which shape style to use for the hatching. These styles are found on the Toolspace Settings tab ➤ General ➤ Multipurpose Styles ➤ Shape Styles.

**See also:**

■ Hatch Tab (Profile View Properties Dialog Box) (page 2182)

**To add hatch areas to a profile view**

**1** Click the profile view grid.

**2** Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties 🔲 .

**3** In the Profile View Properties Dialog Box (page 2178), click the Hatch tab (page 2182).

**4** Use the buttons on the left side to add hatch areas for Cut Area, Fill Area, Multiple Boundaries, or From Criteria.

**5** For each hatch area, review the profiles specified for upper and lower boundaries. If necessary, select different profiles from the list beside each boundary.

**6** Optionally, review the shape styles assigned to the hatch area types, and make any desired changes.

**7** Click OK. The hatch areas are displayed in the profile view.

### Quick Reference

Ribbon

Click profile view. Click Profile tab ➤ Modify View panel ➤ Profile View Properties drop-down ➤ Profile View Properties 🔲

Object Shortcut Menu

Right-click profile view object ➤ Profile View Properties

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <Site name> ➤ Alignments ➤ <Alignment name> ➤ Profile Views, right-click <profile view name> ➤ Properties

Command Line

EditGraphProperties

Dialog Box

Profile View Properties (page 2178)

# Using AutoCAD Object Snaps with Profiles and Profile Views

The following Object Snap modes are supported with profiles and profile views:

| Osnap | Dynamic Profile | Static Profiles | Profile Views |
|---|---|---|---|
| Endpoint | Yes | Yes | Yes (grid lines and axes) |
| Midpoint | Yes | Yes | Yes (grid lines and axes) |
| Center | No | No | No |
| Node | No | No | Yes (tick marks) |
| Quadrant | No | No | No |
| Intersec-tion | Yes | No | Yes (grid lines and axes) |

| Osnap | Dynamic Profile | Static Profiles | Profile Views |
|---|---|---|---|
| Extension | No | No | No |
| Insertion | No | No | No |
| Perpendicular | No | No | No |
| Tangent | No | No | No |
| Nearest | Yes | Yes | Yes (grid lines and axes) |
| Apparent Intersection | No | No | No |
| Parallel | No | No | No |

# Working with Projected Objects

Visualize the location of an object in relation to a surface profile or section.

Projected objects (page 2480) can display their station location and elevation.

**TIP** Before projecting an object into a view, ensure that the elevation of the object has been specified. If the object has a default elevation of 0, the profile view or section view will be vertically expanded to include the 0 elevation.

The following object types can be projected:

- **AutoCAD Civil objects:** Points, feature lines, and survey figures

- **AutoCAD objects:** points, blocks, MView blocks, 3D solids, and 3D polylines

Pipe networks have a separate procedure for projection into profile views. See Displaying Pipe Networks in Profile Views (page 1220) and Displaying Pipe Networks in Section Views (page 1221).

**Editing Projected Objects**

The station location of a projected object always reflects the location of the source object, so it can only be changed by moving the source object.

The style and elevation of projected objects can be edited on the Projections tab of the Profile View Properties (page 2178) or the Section View Properties (page 2281) dialog boxes.

The elevation of a projected object can be set to Use Object. This is a dynamic setting that ensures the elevation of the projected object and the source object are the same. If the source object is edited, the projected object reflects any changes. Elevations can also be set to match a specific surface or profile.

The elevation of some projected objects can be edited using grips. The exceptions are three object types in profile views:

- Survey figures

- Feature lines embedded in a corridor

- Feature lines dynamic to an alignment

Grip editing the vertices of feature lines or 3D polylines in a profile view also changes the elevations of the source objects in plan view. This behavior is useful for editing feature lines in relation to profiles. Grips appear on 3D polylines only if the elevation is set to **Use Object**.

Grip editing point or block objects in profile or section views does not affect the elevations of source objects, but it does break the dynamic link to the source object elevation. The Elevation Options setting changes to Manual. At any time you can change this setting to Use Object, or to match a specific surface or profile.

The object and label styles of a projected object are independent of the styles used by the source object in plan view. You can edit the styles of a projected object without affecting the source object.

**See also:**

- Adding Projected Objects to a Section View (page 1157)
- Editing Section View Properties (page 1156)
- Setting Profile View Properties (page 1042)

# Adding Projected Objects to a Profile View

Project AutoCAD or AutoCAD Civil objects from plan view into a profile view.

You can project AutoCAD points, blocks, 3D solids, and 3D polylines; you can also project AutoCAD Civil COGO points, feature lines, and survey figures.

**To project objects to a profile view**

1 Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Project Objects To Profile View 📊 .

2 Click one or more objects in the drawing that you want to project. Right-click or press Enter after all objects are selected.

3 Click the profile view in which you want the objects to appear.

4 In the Project Objects to Profile View dialog box (page 2192), verify the settings such as Elevation Options and Label Style, then click OK.

## Quick Reference

Ribbon

Click Home tab ➤ Profile & Section Views panel ➤ Profile View drop-down ➤ Project Objects To Profile View 📊

Command Line

ProjectObjectsToProf

Dialog Box

Projections Tab (Profile View Properties Dialog Box) (page 2183)
Project Objects to Profile View Dialog Box (page 2192)

# Editing the Style of a Projected Object

Change the appearance of a projection, without affecting the source object.

Projected AutoCAD Civil objects use standard style editing dialog boxes for points, feature lines, and survey figures. Projected AutoCAD objects use multipurpose styles that control the appearance of different object types in profile views and section views.

**To edit a projection style**

1   In the drawing window, in a profile view or section view, select the object for which to change the style. Right-click. Click the menu option for the object type: Edit Point Style, Edit Feature Line Style, Edit Survey Figure Style, or Edit Projection Style.

    OR

    In Toolspace, on the Settings tab, expand General ➤ Multipurpose Styles ➤ Projection Styles. Right-click the style you want to change. Click Edit.

2   In the Projection Styles dialog box, make changes as required.

## Quick Reference

Ribbon

  Click the projected object. Click Projected Object tab ➤ Modify Projected Object panel ➤ Projection Properties drop-down ➤ Edit Style

Object Shortcut Menu

  Right-click projected object ➤ Edit Projection Style

Toolspace Shortcut Menu

  Settings tab: General ➤ Multipurpose Styles ➤ Projection Styles

Command Line

  EditProjectionStyle

Dialog Box

  Projection Styles Dialog Box (page 1807)

# Removing a Projected Object from a Profile View

Select a projected object and remove it from the profile view.

Other projected objects not selected in the profile view remain in place.

**To remove a projected object**

➤   Click the projected object. Click Projected Object tab ➤ Modify Projected Object panel ➤ Remove From View .

## Quick Reference

Ribbon

  Click the projected object. Click Projected Object tab ➤ Modify Projected Object panel ➤ Remove From View

Command Line

RemoveFromProfileView

# Profile Command Reference

Use commands for quick access to profile functionality.

The following table lists the profile-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AddProfilePVI | Adds a point of vertical intersection (PVI) (page 1093)to an existing profile. |
| CreateProfileFromCorridor | Creates a profile from a corridor feature line (page 1372). |
| CreateProfileFromFile | Creates a profile from a specified text file (page 1050). |
| CreateProfileFromSurface | Opens the Create Profile from Surface dialog box (page 2144), which is the starting point for creating a surface profile from a horizontal alignment. |
| CreateProfileLayout | Creates a layout profile (page 1047) on an existing profile view. |
| CreateProfileReference | Creates a profile reference (page 1055). |
| CreateQuickProfile | Creates a quick profile (page 1058). |
| DeleteProfilePVI | Deletes a specified PVI (page 1093) from a profile line. |
| DesignCriteriaEditor | Opens the Design Criteria Editor dialog box (page 867). |
| EditProfileLabels | For the specified profile, opens the Profile Labels dialog box (page 2167). |
| EditProfileLayout | For the specified profile, opens the Profile Layout Tools dialog box (page 2170). |
| EditProfileProperties | For the specified profile, opens the Profile Properties dialog box (page 2174). |
| SuperimposeProfile | Creates a superimposed profile (page 1054). |

# Profile View Command Reference

Use commands for quick access to profile view functionality.

The following table lists the AutoCAD Civil commands for profile views and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AddProfileViewDepthLbl | Adds a depth label  (page 1037)that shows the spatial relationship between two specified points on a profile view. |
| AddProfileViewLabels | Opens the Add Labels dialog box. (page 1037) |
| AddProfileViewStaElevLbl | Adds a station elevation label (page 1037) to a profile view that shows the station and elevation data for a specified point, which can be anywhere on the profile view grid. |
| CreateMultipleProfileView | Opens the Create Multiple Profile Views Wizard (page 2145). |
| CreateProfileView | Opens the Create Profile View wizard (page 2145). |
| CreateProfileViewStyle | Opens the Profile View Style dialog box (page 2185). |
| EditProfileViewStyle | Opens the Profile View Style dialog box (page 2185) |
| ProjectObjectsToProf | Adds projected objects to a profile view, opening the Project Objects to Profile View Dialog Box (page 2192) dialog box. |
| RemoveFromProfileView | Removes a projected object from a profile view. |

# Sections

# 25

Use sections (also referred to as cross sections) to provide a view of the terrain cut at an angle across a linear feature, such as a proposed road.

Typically, sections are cut across horizontal (plan) alignments at a specified station interval using specified swath widths. These sections are then plotted, individually for a station, or as a group for a range of stations.

AutoCAD Civil handles the creation, management, and plotting of sections with the following components:

■ **Sections**. Terrain elevations that cut across surfaces, including corridor surfaces, which are associated with a specified sample line group. Elevations are sampled at each of the sample line XY vertices and also at locations where the vertical plane defined by the sample line intersects with surface edges.

■ **Section Views**. For each sample line, views displaying some or all of the sections sampled at that sample line. This graphical view has both horizontal limits based on the length of corresponding sample line and vertical values based on the minimum and maximum elevations from the set of sections it is displaying.

**NOTE** Before creating sections, a surface and one or more horizontal alignments must exist.

For more information, see Creating and Editing Sample Lines and Sections (page 1142).

## Understanding Sections

You can use sections to examine existing and proposed surface elevations sampled across an alignment at defined station intervals.

In AutoCAD Civil, section data is defined and displayed using sample lines, sections, and section views, all of which are managed through a collection called a sample line group. An alignment can have more than one sample line group associated with it, each having a unique set of sample lines and sections.

For overview information about AutoCAD Civil objects, see Understanding Objects and Styles (page 51).

## Sample Line Object

Use a sample line as a linear plan object representing the direction along which sections are cut for a set of specified surfaces.
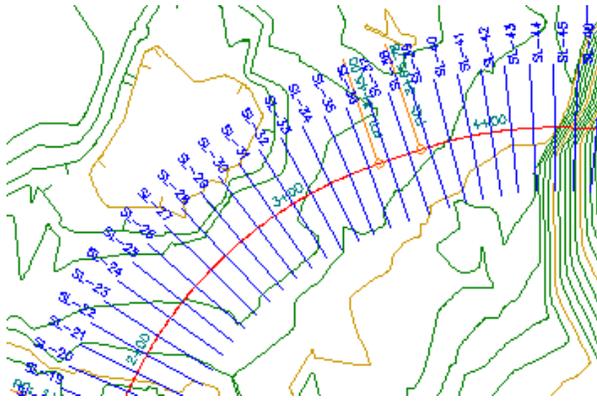
Sample lines are linear objects that are used to cut sections across an alignment. Sample lines have their own styles and can be labeled (for example, SL-29). A set of sample lines makes up a named collection called a sample line group. A sample line group centrally manages the display styles and label styles for a number of sample lines, sections, section views, and mass haul lines and mass haul views.

Sample line groups related to an alignment are displayed in the Toolspace Prospector tree under that alignment.

# Section Object

Use a section object to examine surface elevations along a sample line (across an alignment). In AutoCAD Civil, each section is an object.

Sections cut across a linear feature, to a specified distance to the left and right. Typically, sections are cut at given stations along a horizontal alignment.

A section is an object containing elevation data along a sample line. Each surface that intersects the vertical plane defined by the sample line results in a section object. The type of section object is defined by the elevation source, such as the TIN surface. Sections can also be extracted from corridor surfaces, corridors, and pipe networks along a sample line.

When you create sample lines along an alignment, the section objects are also created for a specified set of existing or proposed surfaces. The default set of surfaces to sample at each sample line are specified through a sample line group. To plot the sections in the drawing, create individual section views or multiple section views.

A section object is defined by the following components:

■ Vertices or grade break points, where a grade line ends and another line might begin

■ Line segments, which detail the grade of the surface between two grade break points

For example, if a section is sampled from a TIN surface, then the grade breaks are extracted at the point where the (vertical plane defined by) sample lines intersect with the TIN edges. Elevation between these two line segments is linearly interpolated and joined with a straight line (assuming that the sample line between these two points is also a straight line).

### Corridor Section Object

The corridor section object is a special type of section object that displays assemblies as applied in a corridor at a specified (corridor) station. While surface-created sections are usually line items with a series of lines joining various grade breaks, corridor-created sections can display closed shapes and multiple layers.

This sectional design is accomplished using subassembly and assembly objects. Subassemblies are design elements that define sectional geometry of a specified part or element used in typical sections. Assemblies

are collections of subassemblies that form shapes of typical sections. These assemblies can be displayed at each station in a section view. Each station is created along the alignment at either a baseline or a controlling offset of that corridor.

For more information, see Assemblies and Subassemblies (page 1377) and Corridors (page 1327).

**Pipe Network Section Object**

The pipe network section object is a special type of section object. It shows pipe networks at a specified station. While surface-created sections are usually line items with a series of lines joining various grade breaks, pipe-created sections can display closed (pipe) shapes.

For more information, see Pipe Networks (page 1185).

# Section View Object

Create section view objects to display sectional data in the drawing.

The section view object displays sectional data (both existing and finished ground/designed) along a sample line, at any given station along a horizontal alignment. A section view shows a section created from either a surface (TIN or corridor), a corridor (which is an assembly at a specified station), or a pipe network. Bands can also be displayed above or below the section view.

A section view, which is very similar to a profile view, is essentially a grid or graph with specific characteristics that are controlled by a section view style. Section views that relate to a sample line are stored in the Prospector tree in Toolspace under that sample line. Individual section views are accessible in Toolspace using the item view. If you delete a sample line in the drawing, section views under that sample line are also deleted.

A section view can include projections of other objects in the drawing that you want to see in relation to a section. See Adding Projected Objects to a Section View (page 1157) for more information.

# Section Object Relationships

Since sample line geometry controls where surfaces are sampled to create section objects, sample lines are the parent of the section object.

You create one or more section views along a sample line to graphically represent section objects at that location across an alignment. Thus, each section and section view are children of a sample line. Each sample line is associated with a sample line group, which is in turn associated with an alignment.

Profiles are longitudinal sections cut along a linear feature and are also children of the parent alignment. For more information, see Profiles (page 1019).

# Sample Line Groups Collection (Prospector Tab)

Use the Sample Line Groups ⊡ collection to access the sample lines, sections, section views, mass haul lines, and mass haul views in a drawing.

As sample line groups are created, they are displayed as named sample line groups, such as SLG-1, in the Sample Line Groups collection.

Sample lines, in turn, are displayed as named sample lines, such as SL-1, in the named sample line groups, which are located in the Sample Line Groups collection:
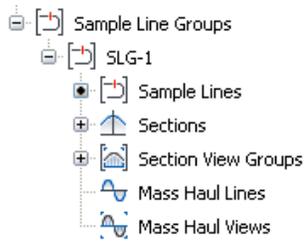
If one or more sample line groups have been added to the current drawing, expand the Sample Line Groups collection to view the names of the sample line groups. A tabular list of the sample line groups is displayed in the item view at the bottom of the Prospector tab.

Select Sample Lines of an individual sample line group to display the sample line group's named sample lines in the item view at the bottom of the Prospector tab. Similarly, expand the subordinate sections, section views, mass haul lines and mass haul views collections to display in the item view the individual sections, section views, mass haul lines and mass haul views. For more information, see The Toolspace Item View (page 82).

Right-click a named sample line group to:

■ Access the sample line group's properties (page 2257).

■ Zoom or pan to the extents of the sample line group.

■ Refresh the view of the Sample Line Groups collection.

## Sections Collection (Prospector Tab)

Use the Sections ⬆ collection to access the sections in a drawing.

As section objects are created, they are displayed as named sections, such as SLG-1-SL-1-EG(1), in the Sections collection for each named sample line.

If one or more sections has been added to the current drawing, expand the Sections collection to view the names of the sections. A tabular list of the various types of sections is displayed in the Prospector list view. These may include:

■ Surface sections 

■ Corridor sections 

■ Corridor surface sections 

■ Pipe network sections 

■ Material list sections 

When each type of section is selected, a tabular list of individual sections of that type is displayed in the item view at the bottom of the Prospector tab. For more information, see The Toolspace Item View (page 82).

## Section View Groups Collection (Prospector Tab)

Use the Section View Groups ⬆ collection to access the section views in a drawing.

As individual section view objects are created, they are displayed as named section views, such as 0+00.00(1), in the item view under Individual Section Views in the Section View Groups collection for each named sample line.

As multiple section view objects are created, they are displayed as named section views, in the item view under their parent section view group in the Section View Groups collection.

For more information, see The Toolspace Item View (page 82).

## Sample Line Collection (Settings Tab)

Use this collection in the Toolspace Settings tree to manage sample line settings, sample line styles, sample line label styles, and sample line command settings.

Right-click the Sample Line collection to:

■ Edit the sample line feature settings

■ Edit the sample line label style defaults

■ Refresh the display of the sample line settings collection

For more information about the contents of this shortcut menu, see "The Object Collection (Settings Tree)".

Expand the Sample Line collection to display and edit the styles and command settings that are available for sample lines.

| For information about... | See... |
| --- | --- |
| Sample line settings | Sample Line, Section, and Section View Settings (page 1128) |
| Sample line styles | Sample Line, Section, and Section View Styles (page 1131) |
| Sample line label styles | Using Sample Line Labels (page 1136) |
| Sample line commands | Sections Command Reference (page 1158) |

## Section Collection (Settings Tab)

Use this collection in Toolspace in the Settings tree to manage section settings, section styles, and section label styles.

Right-click the Section collection to:

■ Edit the section feature settings

■ Edit the section label style defaults

■ Refresh the display of the sample line settings collection

For more information about the contents of this shortcut menu, see The Object Collection (Settings Tree) (page 93).

| For information about... | See... |
| --- | --- |
| Section settings | Section Settings (page 1129) |

| For information about... | See... |
| --- | --- |
| Section styles | Creating and Editing Section Styles (page 1132) |
| Section label styles | Using Section Labels (page 1137) |

## Section View Collection (Settings Tab)

Use this collection in Toolspace in the Settings tree to manage section view settings, section view styles, section view label styles, data band styles, group plot styles, sheet styles, and section view command settings.

Right-click the Section View collection to:

■ Edit the section view feature settings

■ Edit the section view label style defaults

■ Refresh the display of the sample line settings collection

For more information about the contents of this shortcut menu, see The Object Collection (Settings Tree) (page 93).

| For information about... | See... |
| --- | --- |
| Section view settings | Section View Settings (page 1130) |
| Section view styles | Creating and Editing Section View Styles (page 1133) |
| Section view label styles | Using Section View Labels (page 1139) |
| Band styles | Creating and Editing Section View Styles (page 1133) |
| Sheet styles | Creating and Editing Section View Styles (page 1133) |
| Group plot styles | Creating and Editing Section View Styles (page 1133) |
| Section view commands | Sections Command Reference (page 1158) |

# Sample Line, Section, and Section View Settings

You can change the default style and name format settings for sample lines, sections, and section views.

Settings are handled in a standard way throughout AutoCAD Civil via the Settings tree at three levels: drawing, feature, and command. For more information, see Understanding Settings (page 61).

## Sample Line Settings

To establish default settings for all sample lines, in the Settings tree, right-click the Sample Line collection.

To change settings for the CreateSampleLine command, expand the Commands collection in the Sample Line collection and right-click the CreateSampleLine command.

The default styles and name format settings for sample line commands are described in this topic. Drawing ambient settings are described in Specifying Drawing Settings (page 63).

**To change sample line settings**

1  In Toolspace, on the Settings tab, right-click the Sample Line collection. Click Edit Feature Settings.

   In the Edit Feature Setting - Sample Line dialog box, note that appears next to the settings for default styles and default name format, marking them as specific to sample lines.

2  Expand the default styles and the default name format to see the current settings.

3  In the Value column, click the entry you want to change.

4  Click to open a dialog box. Modify the value.

5  In the Edit Feature Settings - Sample Line dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

   Settings tab: right-click Sample Line collection ➤ Edit Feature Settings

Dialog Box

   Edit Feature Settings - Sample Line (page 2243)

# Section Settings

You can change the default style and name format settings for sections.

To establish default settings for all sections, in the Settings tree, right-click the Section collection.

The default styles and name format settings for sections are described in this topic. Drawing ambient settings are described in Specifying Drawing Settings (page 63).

**To change section settings**

1  In Toolspace, in the Settings tree, right-click the Section collection. Click Edit Feature Settings.

   In the Edit Feature Settings - Section dialog box, note that appears next to the settings for default styles and the default name format, marking them as specific to sections.

2  Expand the default styles and the default name format to see the current settings.

3  In the Value column, click the entry you want to change.

4  Click to open a dialog box. Modify the value.

5  In the Edit Feature Settings - Section dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Section collection ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings - Section (page 2245)

## Section View Settings

You can change the default style and name format settings for section views and section view commands.

To establish default settings for all section views, in the Settings tree, right-click the Section View collection.

To change settings for a specific command, use the Commands collection in the Section View collection.

The default styles and name format settings for section view commands are described in this topic. Drawing ambient settings are described in Specifying Drawing Settings (page 63).

---

**NOTE** Overrides to the drawing ambient settings at the Section View collection level and the Command collection level affect only the specified level. The drawing level settings are not changed. For more information, see The Commands Collections (Settings Tree) (page 97).

---

**To change section view settings**

1   In Toolspace, in the Settings tree, right-click the Section View collection. Click Edit Feature Settings.

   In the Edit Feature Settings - Section View dialog box, note that 🏠 appears next to the settings for the default styles and the default name format, marking them as specific to section views.

2   Expand the default styles and the default name format to see the current settings.

3   In the Value column, click the entry you want to change.

4   Click ⊡ to open a dialog box. Modify the value.

5   In the Edit Feature Settings - Section View dialog box, click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Section View collection ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings - Section View (page 2246)

## Sample Line, Section, and Section View Styles and Display

Styles control the way a sample line, section, and section view is displayed in a drawing. Sample lines, sections, and sections views have label styles that control the appearance and behavior of their labels in a drawing.

# Sample Line, Section, and Section View Styles

Use styles to control how sample lines, sections, and section views are displayed in a drawing.

In AutoCAD Civil, all objects have a standard object style grouping on the Settings tree, called an object style collection. You use this to create, edit, copy, and delete the styles for that object. For more information, see The Object Style Collection (Settings Tree) (page 94).

## Creating and Editing Sample Line Styles

Use the Toolspace Settings tree to create, copy, edit, or delete a sample line style.

Create a new style by copying an existing style. Then change its properties to suit your requirements. Use the Sample Line Style dialog box (page 2264) tabs to specify the main properties.

---

**NOTE** Before editing an existing sample line style to change its properties, remember that any changes you make will apply to all sample lines that use the style. If you want to change the style of some sample lines but not all, consider creating a new style.

---

**To create a new sample line style**

1   In Toolspace, in the Settings tree, expand the Sample Line Styles collection.

2   Do one of the following:
    ■   Right-click Sample Line Styles collection. Click New.

    ■   Right-click the existing style to use as a template. Click Copy.

3   In the Sample Line Style dialog box (page 2264), enter or change the name of the style and other settings as required.

4   Click OK.

**To copy an existing sample line style**

1   In Toolspace, in the Settings tree, expand the Sample Line Styles collection.

2   Right-click the existing style. Click Copy.

3   In the Sample Line Style dialog box (page 2264), change the name of the style and other settings as required.

4   Click OK.

**To edit a sample line style**

1   In Toolspace, in the Settings tree, expand the Sample Line Styles collection.

2   Right-click the existing style. Click Edit.

3   In the Sample Line Style dialog box (page 2264), change settings as required.

4   Click OK.

**To delete a sample line style**

1   In Toolspace, in the Settings tree, expand the Sample Line Styles collection.

2   Right-click the style to delete. Click Delete.

3   In the confirmation dialog box, click Yes.

### Quick Reference

Toolspace Shortcut Menu

Create: Settings tab, right-click Sample Line Styles collection ➤ New
Edit: Settings tab, Sample Line Styles collection, right-click <style name> ➤ Edit

Command Line

CreateSampleLineStyle
EditSampleLineStyle

Dialog Box

Sample Line Style (page 2264)

## Creating and Editing Section Styles

Use the Toolspace Settings tree to create, copy, edit, or delete a section style.

Create an entirely new style, or create a new style by copying an existing style. Change its properties to suit your requirements. Use the Section Style dialog box (page 2274) tabs to specify the main properties.

---

**NOTE** Before editing an existing section style to change its properties, remember that any changes you make will apply to all sections that use the style. To change the style of some sections, but not all, consider creating a new style.

---

**To create a new section style**

1  In Toolspace, in the Settings tree, expand the Section Styles collection.

2  Do one of the following:
   ■ Right-click Section Styles collection. Click New.

   ■ Right-click the existing style you want to use as a template. Click Copy.

3  In the Section Style dialog box (page 2274), change the name of the style and other settings as required.

4  Click OK.

**To copy an existing section style**

1  In Toolspace, in the Settings tree, expand the Section Styles collection.

2  Right-click the existing style. Click Copy.

3  In the Section Style dialog box (page 2274), change the name of the style and other settings as required.

4  Click OK.

**To edit a section style**

1  In Toolspace, in the Settings tree, expand the Section Styles collection.

2  Right-click the existing style. Click Edit.

3  In the Section Style dialog box (page 2274), change settings as required.

4  Click OK.

**To delete a section style**

1   In Toolspace, in the Settings tree, expand the Section Styles collection.

2   Right-click the existing style to delete. Click Delete.

3   In the confirmation dialog box, click Yes.

## Quick Reference

Toolspace Shortcut Menu

Create: Settings tab, right-click Section Styles collection ➤ New
Edit: Settings tab, Section Styles collection, right-click <style name> ➤ Edit
Delete: Settings tab, Section Styles collection, right-click <style name> ➤ Delete

Command Line

CreateCrossSectionStyle
EditCrossSectionStyle

Dialog Box

Section Style (page 2274)

## Creating and Editing Section View Styles

Use the Toolspace Settings tree to create, copy, edit, or delete a section view style.

Create an entirely new style, or create a new style by copying an existing style. Then change its properties to suit your requirements. Use the Section View Style dialog box (page 2288) tabs to specify the main properties.

---

**NOTE** Before editing an existing section view style to change its properties, remember that any changes you make will apply to all section views that use the style. To change the style of some section views, but not all, consider creating a new style.

---

Section view styles control the format of the graph on which the sections are displayed, as well as the title and annotations on the axes.

Label styles control the format of labels for offset elevations at any point in a section view and depth/grade between any two selected points in a section view (see Using Section Labels (page 1137)).

Band styles control the format of section bands, which can appear above and/or below the section view.

For convenience, define a group of band styles as a band set. The entire set of band styles can then be applied to a section view with a single step, rather than applying each band style separately.

Group plot styles control the layout of the graph on which multiple section views are displayed. It also controls the spacing between sheets (if plotted "by page") and overall plot area.

For more information, see Group Plot Style dialog box. (page 2251)

Sheet styles control the way a cross section sheet, which contains multiple section views, is displayed. They also specify the individual plot area and visibility of sheet components.

For more information, see Sheet Style Dialog Box (page 2296).

Projection styles control the appearance of AutoCAD point, solid, polyline, and block objects projected into profile views or section views.

**To create a new section view style**

**1** In Toolspace, in the Settings tree, expand the Section View Styles collection.

**2** Do one of the following:
- Right-click Section View Styles collection. Click New.
- Right-click the existing style to use as a template. Click Copy.

**3** In the Section View Style dialog box (page 2288), enter or change the name of the style and other settings as required.

**4** Click OK.

**To copy an existing section view style**

**1** In Toolspace, in the Settings tree, expand the Section View Styles collection.

**2** Right-click the existing style. Click Copy.

**3** In the Section View Style dialog box (page 2288), change the name of the style and other settings as required.

**4** Click OK.

**To edit a section view style**

**1** In Toolspace, in the Settings tree, expand the Section View Styles collection.

**2** Right-click the existing style. Click Edit.

**3** In the Section View Style dialog box (page 2288), change settings as required.

**4** Click OK.

**To delete a section view style**

**1** In Toolspace, in the Settings tree, expand the Section View Styles collection.

**2** Right-click the existing style to delete. Click Delete.

**3** In the confirmation dialog box, click Yes.

**To edit a section view group plot style**

**1** Select the section view whose group plot style you want to edit.

**2** Click Section tab ➤ Modify View panel ➤ View Group Properties drop-down ➤ Edit Group Plot Style .

**3** In the Group Plot Style dialog box (page 2251), change settings as required.

**4** Click OK.

**To update a section view group's layout**

**1** Select a section view whose group layout may be modified by a style change.

**2** Click Section tab ➤ Modify View panel ➤ Update Group Layout .
The layout of the section view group is updated to reflect any style changes.

## Quick Reference

## Creating and Copying Projection Styles

Use these general styles to control the display of AutoCAD objects projected into profile views and section views.

Within a style, the settings for the appearance of an object type in section views are independent from the settings for profile views.

**To create a projection style**

1   In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Right-click Projection Styles and click New.

2   In the Projection Styles dialog box (page 1807), specify the name of the style and other settings as required.

3   Click OK.

**To copy a projection style**

1   In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Identify the style you want to copy as a basis for the new style.

2   Right-click the style name and click Copy.

3   In the Projection Styles dialog box (page 1807), change the name of the style and other settings as required.

4   Click OK.

## Quick Reference

## Editing Projected Object Styles

Change the appearance of a projected object, without affecting the source object.

Projected AutoCAD Civil objects use standard style editing dialog boxes for points, feature lines, and survey figures. Projected AutoCAD objects use multipurpose styles that control the appearance of different object types in profile views and section views.

**To edit a projection style**

1 In Toolspace, on the Settings tab, expand the General ➤ Multipurpose Styles collection. Identify the style you want to edit.

2 Right-click the style name and click Edit.

3 In the Projection Styles dialog box (page 1807), change settings as required.

4 Click OK.

**To edit the style of a projected object**

1 In the drawing window, in a section view, select the object whose style you want to change.

2 Right-click, and click the menu option for the object type:
   - Edit Point Style
   - Edit Feature Line Style
   - Edit Survey Figure Style
   - Edit Projection Style

3 In the styles dialog box, make changes as required.

## Quick Reference

Object Shortcut Menu

Right-click projected object ➤ Edit Projection Style

Toolspace Shortcut Menu

Settings tab: General ➤ Multipurpose Styles ➤ Projection Styles

Dialog Box

Projection Style (page 1807)

# Sample Line, Section, and Section View Labels

You can use sample line, section, and section view label styles to control the appearance and behavior of labels in a drawing.

This information describes what is unique to sample line, section, and section view labels. For a general overview of labels, see Labels and Tags (page 1463).

## Using Sample Line Labels

Use the Toolspace Settings tree to create and edit sample line label styles, which are used to label sample line objects.

Specify which label styles to use when you create sample lines.

**To add sample line labels automatically based on the default label style**

**1** In the Toolspace Settings tree, right-click Sample Line and click Edit Feature Settings.

**2** In the Edit Feature Settings - Sample Line dialog box (page 2243) click Default Styles ➤ Sample Line Label Style.

**3** Click in the Value column and click ⊡ .

**4** In the Sample Line Label Style dialog box (page 1805), set the default label style.

Sample line labels are displayed when you add sample lines to the drawing.

You can select a sample line label style in the Sample Line Labels dialog box, or create or edit a style in the Label Style Composer.

**To edit sample line labels**

**1** Select a sample line in the drawing.

**2** Click Sample Line tab ➤ Labels & Tables panel ➤ Edit Labels ✎ .

**3** In the drawing, select a sample line to open the Sample Line Labels dialog box (page 2261).

**4** Edit the label in the Label Style Composer dialog box (page 1942).

## Quick Reference

Ribbon

Select the sample line. Sample Line tab ➤ Labels & Tables panel ➤ Edit Labels ✎

Menu

Sections menu ➤ Add Section View Labels ➤ Add/Edit Sample Line Group Labels

Object Shortcut Menu

Right-click a sample line ➤ Edit Sample Line Labels in Group

Command Line

EditSampleLineLabels

Dialog Boxes

Sample Line Labels (page 2261)
Label Style Composer (page 1942)

## Using Section Labels

Use the Toolspace Settings tree to create and edit section label styles.

When you create sections, specify which label styles to use. Sections can be labeled automatically when created, using the specified label set. After creating a section, you can also add labels.

Use either the Section Labels dialog box (page 2270) or the Section View Properties dialog box (page 2281)to add labels.

You can select a section label style in the Section Labels or Section View Properties dialog box, or create or edit a style in the in the Label Style Composer.

Types of section label styles:

| This type of section label style... | Labels... |
| --- | --- |
| Major Offset | Offsets, elevations, and instantaneous grades at major increments. |
| Minor Offset | Offsets, elevations, and instantaneous grades at minor increments. |
| Segments | Section segment labels for each segment (line) of a section object. A user-specified weeding factor is supported, thus avoiding close annotation. |
| Grade Breaks | Grade break labels applied at every grade break point for a section. A user-specified weeding factor is supported, thus avoiding close annotation. |
| Label Set | The default set of label styles to use for sections when a section view object is created. |

For a general overview of labels, see Labels and Tags (page 1463).

**To add section labels**

1  In the drawing, zoom to a section view.

2  Do one of the following:
   ■ Click the desired section line. Right-click and click Edit Labels.
   ■ Click the desired section view grid. Right-click and click Section View Properties. On the Sections tab, click the Labels column.

   The Section Labels dialog box (page 2270) is displayed.

3  In the Section Labels dialog box, for Type, select the label type in the list, either: Major Offset, Minor Offset, Segments, or Grade Breaks.

4  Optionally, for Section <Label Style> Label Style, select the label style in the list or use the standard controls to create a new style, copy or edit the current style selection, create a child of the current style selection, or pick a style from the drawing.

5  Click Add.
   The specified label type is added to the set.

6  Review the label set in the table and make the required adjustments. Click OK.
   The labels are displayed along the section line in the section view.

**To edit section labels**

1  Click the section line you want to edit.

2  Do one of the following:
   ■ Click Section tab ➤ Labels panel ➤ Edit Section Labels 🖋 .
   ■ Right-click and click Edit Labels.

**3** In the Section Labels dialog box (page 2270), review the existing labels and their attributes. Do any of the following:

- Add or remove labels of any type.

- Change the style for a label type.

- Import a standard set of labels.

- Save your modified labels as a set to use again.

- Stagger labels to avoid label collisions. For more information, see Staggering Profile and Section Labels (page 1527).

**4** Click Apply to see your changes on the section line.

**5** Click OK to save the changes.

## Quick Reference

Ribbon

Select the section line. Section tab ➤ Labels panel ➤ Edit Section Labels 

Object Shortcut Menu

Click section, right-click ➤ Edit Labels

Command Line

EditSectionLabels

Dialog Box

Section Labels (page 2270)

## Using Section View Labels

Use the Toolspace Settings tree to create and edit section view label styles.

You can label section views automatically when you create them, using specified label styles. After creating section views, you can also add Offset Elevation and Grade labels.

If a section view contains projected AutoCAD objects, label styles for these objects are managed as feature settings for the section view object.

Projection labels show the location of an object projected into the section view, such as a block or a point.

Use the Add Section View Labels option to access the Add Labels dialog box (page 1970) if you need to create or modify a style before labeling a section view.

| This type of section view label style... | Labels... |
| --- | --- |
| Offset Elevation | The elevation and offset value at a specified point in the section view, not necessarily on a section. |
| Grade | The elevation difference between two specified points, as well as other selected data, such as slope and distance between the points. |

**To add a section view grade label**

1  Zoom to, and select, the section view that you want to label.

2  Click Section View tab ➤ Labels panel ➤ 🏷 Add View Labels drop-down ➤ 🏷 Grade.

3  Click the section view grid that you want to label.

4  Click any two points. A label is displayed, showing the grade label (default) or other data about the relationship between the two points (depending on the label style).

5  Repeat Step 4 for each point or pair of points that you want labeled.

6  To end the procedure, press Esc. Close the Add Labels dialog box.

**To add a section view offset elevation label**

1  Zoom to, and select, the section view that you want to label.

2  Click Section View tab ➤ Labels panel ➤ 🏷 Add View Labels drop-down ➤ 🏷 Offset Elevation.

3  Click the section view grid that you want to label.

4  Click the location where you want a label. It is immediately displayed, showing the offset and elevation values (default).

5  Repeat Step 4 for each point or pair of points that you want labeled.

6  To end the procedure, press Esc. Close the Add Labels dialog box.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Section View ➤ Grade 🏷

Select the section view. Section View tab ➤ Labels panel ➤ 🏷 Add View Labels drop-down ➤ 🏷 Grade

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Section View ➤ Offset Elevation 🏷

Section View tab ➤ Labels panel ➤ 🏷 Add View Labels drop-down ➤ 🏷 Offset Elevation

Menu

Sections menu ➤ Add Section View Labels ➤ Grade
Sections menu ➤ Add Section View Labels ➤ Offset Elevation

Command Line

AddSectionViewGradeLbl
AddSectionViewOffElevLbl

# Section View Bands

You can use section view bands to control the location and content of bands in section views.

A data band is an optional graphic frame that is associated with a section view. The data band contains annotations for the section view and the section objects displayed in that section view. Some common annotations include elevation data, offsets, and cut/fill depths.

Bands can be created automatically when you create section views and also added manually to a section view you already created. Use the Section View Properties dialog box.

**Section View Band Styles**

Use the Toolspace Settings tree to create and edit section view band styles. Specify which styles to use when you create section views. Add labels using the Add Labels dialog box (page 1970).

Data band labels are label objects that can be manipulated like other AutoCAD Civil label objects. For more information, see Editing Data Band Labels (page 1529).

Group a set of commonly used bands into a band set that can be applied to a section view as a single selection.

| This type of band style... | Labels... |
| --- | --- |
| Band Sets | Specifies the default set of bands to use (or import) when creating a section view. Also, specifies various properties, such as the gap between bands. |
| Section Data | Specifies the offsets from centerline, elevations of any section, and the cut/fill depths at major/minor increments and other points of interest along the sample line. |
| Section Segment | Specifies section segment band style for each segment (line) of a section object. |

# Adding Section View Bands

Add data bands that display data such as elevations and horizontal geometry.

Place data bands along the top or bottom of a section view grid to annotate the various section lines with elevation data, vertical or horizontal geometry points, superelevation, or pipe network data. You can also group a set of commonly used data bands into a band set that can be applied to a section view as a single selection.

**See also:**

- Section View Band Styles (page 1141)
- Section View Band Set Dialog Box (page 2275)
- Staggering Data Band Labels (page 1528)

**To add bands to a section view**

1  In the section view, click the section view grid.

2  Right-click the section view. Click Section View Properties.

3  In the Section View Properties dialog box, click the Bands tab (page 2283).

4  In the List Of Bands area, select either Bottom or Top as the location for the new band in the section view.

5  Select a band type and style. Click Add.

6  To add more bands to the section view, repeat Steps 4 and 5.

7  Optionally, on the right side of the List Of Bands, move selected bands up ⇧ or down ⇩ in the stack. To delete any unwanted bands, click ✖ .

8  To save this arrangement of bands as a set for reuse, click Save As Band Set.

9  Click OK to close the Section View Properties dialog box. The bands are displayed in the section view.

**NOTE** To label corridor lane widths, create a corridor, build a corridor surface using ETW (edge-of-traveled-way) feature lines, and label that surface with an appropriate section segment band. For more information, see Creating and Editing Section Views (page 1152) and Understanding Corridor Modeling (page 1327).

### Quick Reference

Shortcut Menu

Right-click a selected section view ➤ Section View Properties

Dialog Box

Section View Properties - Bands tab (page 2283)

# Creating and Editing Sample Lines and Sections

AutoCAD Civil enables you can to easily create, manage, and edit sample lines and sections.

# Creating Sample Lines

Create sample lines along an existing horizontal alignment.

Sections are cut along each of the samples lines for a specified set of surfaces. Corridor sections are created when you sample a corridor.

You can create one or more sample lines that are associated with an alignment. In Toolspace, in the Prospector tree, the sample lines are children of an alignment and appear in the hierarchy in the Sample Line Groups collection.

**NOTE** When you create sample lines for the first time for an alignment, you are prompted to create a sample line group in which to place the sample lines. This group also contains the list of surfaces, corridors, corridor surfaces, and pipe networks (if any), based on the alignment so that you can select the data source(s) to be sampled.

In the drawing, sample lines can be labeled with annotations, including stations and other information.

The section can be displayed using the section view. For more information, see Creating and Editing Section Views (page 1152).

**To create a perpendicular sample line, of a specified swath width, across an alignment at a specified station**

1   Click Home tab ➤ Profile & Section Views panel ➤ Sample Lines ⬚ .
    The cursor changes to pick mode. You are prompted to select an alignment.

2   Do one of the following:
    ■ Press Enter to select from a list of alignments.

    ■ In the drawing, click the desired horizontal alignment.

    For information about creating an alignment, see Creating Alignments (page 887).

3   Specify a sample line group. A sample line must belong to a sample line group. Do one of the following:
    ■ If this is the first sample line group you are creating in the drawing, the Create/Edit Sample Line Group dialog box (page 2231) is automatically displayed. Review the default styles. Change them if necessary. Click OK.

    ■ If a sample line group already exists in the drawing, the Sample Line Tools toolbar (page 2265) is displayed. To assign a sample line group to the sample line, select the sample line group in the list.

The cursor changes to station pick mode (default). You are prompted to select a station.

**4** Do one of the following:

- At the command line, specify a station by entering the desired station value.

- In the drawing, use the cursor tooltip to find the station along the alignment. Click the station.

**5** Do one of the following:

- At the command line, enter the left swath width. Press Enter.

- In the drawing, at the desired distance, click the first point, which defines the start of the left swath width. Click the second point, which defines the end of the left swath width (that is, the length).

**6** Do one of the following:

- At the command line, enter the right swath width. Press Enter.

- In the drawing, at the desired distance, click the first point, which defines the start of the right swath width. Click the second point, which defines the end of the right swath width (that is, the length).

**7** Right-click to end the By Stations mode.

Depending on the sample line style, a line is drawn perpendicular across the alignment at the specified location and length. This sample line represents the section.

---

**NOTE** For left and right swath widths, you can attach an alignment, such as a right-of-way alignment. This enables you to define variable sample line widths. If you later use the Edit Swath Widths for Group tool on the Sample Line Tools toolbar (page 2265), all sample line widths will be extended or trimmed to the group's widths.

---

**To create a sample line or multi-segment sample line by picking points on-screen**

**1** Click Home tab ➤ Profile & Section Views panel ➤ Sample Lines ⌐⁂ .

The cursor changes to pick mode. You are prompted to select an alignment.

**2** Do one of the following:

- Press Enter to select from a list of alignments.

- In the drawing, click the desired horizontal alignment.

For more information about creating an alignment, see Creating Alignments (page 887).

**3** Specify a sample line group. A sample line must belong to a sample line group. Do one of the following:
- If this is the first sample line group you are creating in the drawing, the Create/Edit Sample Line Group dialog box (page 2231) is automatically displayed. Review the default styles. Change them if necessary. Click OK.

- If a sample line group already exists in the drawing, the Sample Line Tools toolbar (page 2265) is displayed. To assign a sample line group to the sample line, select the sample line group in the list.

The cursor changes to station pick mode (default). You are prompted to select a station.

**4** On the Sample Line Tools toolbar, in the sample line create method list, click ⌐ , which selects Pick Points On Screen. You are prompted to select a start point.

**5** In the drawing, use the cursor tooltip to find the start point. Click the start point.

**6** Do one of the following:

- In the drawing, click an end point on the opposite side of the alignment. Finish the pick point session by right-clicking. Press Enter.

■ In the drawing, click one or more points to define the sample line (section) with the final end point on the opposite side of the alignment from the start point. Note that if the final end point is not on the opposite side of the alignment from the start point, the sample line is interpolated to attach to the centerline of the alignment.

7 Do one of the following:

■ Continue picking points on screen to draw more sample lines.

■ Right-click to end the Pick Points On Screen mode.

---

**NOTE** When you create a sample line that is located between corridor stations, the data shown in the section is interpolated between the preceding and succeeding stations.

---

Depending on the sample line style, a line or multi-segment line is drawn across the alignment that represents the section.

**To create perpendicular sample lines, by a range of stations, across an alignment at specified station intervals**

1 Click Home tab ➤ Profile & Section Views panel ➤ Sample Lines ⬚ .
The cursor changes to pick mode. You are prompted to select an alignment.

2 Do one of the following:

■ Press Enter to select from a list of alignments.

■ In the drawing, click the desired horizontal alignment.

For information about creating an alignment, see Creating Alignments (page 887).

3 Specify a sample line group. A sample line must belong to a sample line group. Do one of the following:

■ If this is the first sample line group you are creating in the drawing, the Create/Edit Sample Line Group dialog box (page 2231) is automatically displayed. Review the default styles. Change them if necessary. Click OK.

■ If a sample line group already exists in the drawing, the Sample Line Tools toolbar (page 2265) is displayed. To assign a sample line group to the sample line, select the desired sample line group in the list.

The cursor changes to station pick mode (default). You are prompted to select a station.

4 On the Sample Line Tools toolbar, in the sample line create method list, click 🖈 , which selects By Station Range.
The Create Sample Lines - By Station Range dialog box (page 2236)is displayed.

5 Review the default settings. Change them if necessary. Click OK. Note that the station range defaults to the entire length of the alignment.
Depending on the sample line style, sample lines are drawn along the alignment, cutting across the alignment at the specified intervals. These sample lines represent the sections.

**To create perpendicular sample lines across an alignment at specified corridor stations**

1 Click Home tab ➤ Profile & Section Views panel ➤ Sample Lines ⬚ .
The cursor changes to pick mode. you are prompted to select an alignment.

2 Do one of the following:

■ Press Enter to select from a list of alignments.

■ In the drawing, click the desired horizontal alignment.

For more information about creating an alignment, see Creating Alignments (page 887).

3 Specify a sample line group. A sample line must belong to a sample line group. Do one of the following:

■ If this is the first sample line group you are creating in the drawing, the Create/Edit Sample Line Group dialog box (page 2231) is displayed. Review the default styles. Change them if necessary. Click OK.

■ If a sample line group already exists in the drawing, the Sample Line Tools toolbar (page 2265) is displayed. To assign a sample line group to the sample line, select the desired sample line group in the list.

The cursor changes to station pick mode (default). You are prompted to select a station.

4 On the Sample Line Tools toolbar, in the sample line create method list, click ▦ to select From Corridor Stations.

The Create Sample Lines - From Corridor Stations dialog box (page 2236) is displayed.

**NOTE** If you do not have any corridors defined based on the selected alignment, then this command will be inactive.

5 Review the default settings. Change them if necessary. Click OK. Note that the station range defaults to the entire length of the alignment.

Depending on the sample line style, sample lines are drawn along the alignment, cutting across the alignment at the specified corridor stations. These sample lines represent the corridor sections.

**NOTE** For left and right swath widths, you can attach an alignment, such as a right-of-way alignment. This enables you to define variable sample line widths. If you later use the Edit Swath Widths for Group tool on the Sample Line Tools toolbar (page 2265), all sample line widths will be extended or trimmed to the group's widths.

### Quick Reference

Ribbon

Home tab ➤ Profile & Section Views panel ➤ Sample Lines ▦

Menu

Sections menu ➤ Create Sample Lines

Command Line

CreateSampleLines

# Editing Sample Lines

Edit sample lines to change their parameters, move, copy, delete, or replace them, and change their appearance.

Use the editing functionality in the Edit Sample Line dialog box (page 2250) to change the name, length, vertex locations, and station value parameters.

In the drawing, use sample line grips to specify the stretch point, base point, or to copy sample lines. You use the grips to change swath widths or stations.

**To use parameters to edit sample line location and length**

**1** In the drawing, click the sample line you want to edit.

**2** Click Sample Line tab ➤ Modify panel ➤ Edit Sample Line 🖉 .
Note that the Edit Sample Line dialog box and Sample Line Tools toolbar are both displayed.

**3** In the Edit Sample Line dialog box (page 2250), review the parameters for each vertex of the sample line by clicking Previous Vertex and Next Vertex.

**4** At the vertex you want to edit, do one or more of the following:

■ In the Name field, change the name of the sample line.

■ In the Sample Line Vertex Northing and/or Sample Line Vertex Easting field(s), enter new values.

■ In the Sample Line Segment Length field, change the length of the swath width to the desired distance in ground units or click 🔲 to pick a new segment length from the drawing.

■ In the Sample Line Station Value field, change the sample line station value or click 🔳 to pick a new station value from the drawing.

**5** Close the Edit Sample Line dialog box and Sample Line Tools toolbar.
The sample line is redrawn with the new parameters.

**To use grips to edit a sample line**

**1** In the drawing, click the sample line you want to edit.
The sample line displays grip edit locations at each vertex.

**2** Use the diamond-shaped center grip to move the sample line to a new location along the alignment.
The sample line slides to the new location while maintaining its relative geometry. As the sample line moves, it reorients to the underlying entity (for example, spiral, curve, or other tangents). It always remains perpendicular to the alignment.

**3** Use the square-shaped end grips to stretch (lengthen or shorten), move, or rotate segments.
The sample line is redrawn.

**4** Use the triangular-shaped grips at the ends of segments to stretch (lengthen or shorten) each segment while keeping its relative direction.
The sample line is redrawn.

**5** Press Esc or enter eXit to complete the procedure.
The sample line segments are moved or edited.

**To restore a rotated sample line to the perpendicular**

**1** In the drawing, click a sample line.
The sample line displays grip edit locations at each vertex.

**2** Use the square-shaped end grips to rotate segments off their perpendicular orientation to the parent alignment.
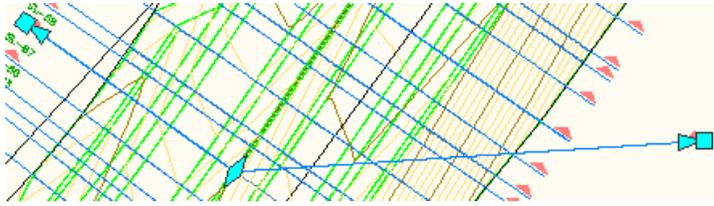
**3** Click the rotated sample line.

**4** Do one of the following:

- Right-click and click Make Orthogonal.

- On the command line, enter MakeSampleLineOrthogonal.

The rotated sample line (1-2) rotates back to a perpendicular orientation (3) to its parent alignment, but retains the length to which it was stretched or contracted.
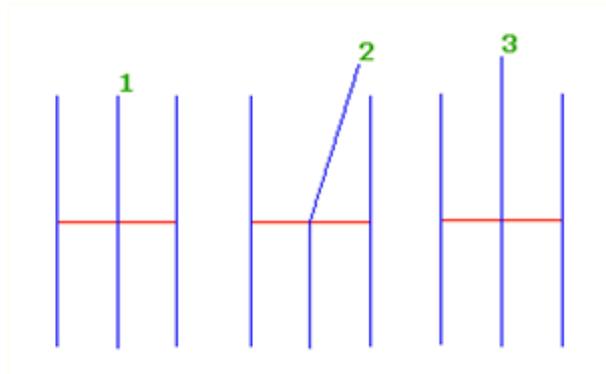


**To change the display style of a sample line**

**1** In the drawing, click the sample line.

**2** Right-click the sample line. Click Sample Line Properties.

**3** In the Sample Line Properties dialog box, click the Information tab (page 2262).

**4** In the Object Style field, select a different style in the list.

**To resample section sources**

**1** Select a sample line in the drawing.

**2** Click Sample Line tab ➤ Modify panel ➤ Sample More Sources 🔺 .

**3** In the Section Sources dialog box, do one or more of the following:

- Select a source in the Available Sources field and click Add >> to add it to the Sampled Sources field.

- Select a source in the Sampled Sources field and click Remove << to remove it From the Available Sources field.

**4** In the Section Sources dialog box, click OK.

### Quick Reference

Ribbon

Sample Line tab ➤ Modify panel ➤ Edit Sample Line ![icon]

Sample Line tab ➤ Modify panel ➤ Sample More Sources ![icon]

Menu

Sections menu ➤ Edit Sample Lines

Command Line

EditSampleLine
SampleSectionSources
MakeSampleLineOrthogonal

# Editing Sections

Edit a section to change its appearance or move it in the drawing.

Use the section editing functionality within a section view. For example, you can move a section in the vertical plane within a section view.

You can use section grip editing to manipulate static sections (that is, sections that are not dynamic). For example, you can alter the section yet maintain the grade or elevation while stretching a segment or moving grade points.

If the section is dynamic, the data is grayed out and not editable. Change the Update Mode to Static in the Section Properties dialog box if you want to edit a section.

For more information about static and dynamic update modes for sections, see Section Data Tab (Section Properties Dialog Box) (page 2272).

**To edit and move a section**

1   In the drawing, zoom to a section view. Click the static section line you want to edit.

    The section line is displayed with triangular grips at the start and end of each section segment and rectangular grips at grade breaks.

2   Move the section segments. Use rectangular grips for grade points. Use triangular grips for segment stretching (lengthen or shorten).

    As you move the grips, the X and Y values are displayed dynamically.

    ---
    **NOTE** To edit the section segment by editing the dynamic dimensions in place, press Tab to cycle through the offset and elevation values displayed.

    ---

3   Click in the drawing at the desired location.

    The section is redrawn at the new location.

    ---
    **NOTE** The section must be a static type section (that is, a section that is not dynamic). For more information about static and dynamic update modes for sections, see Section Data Tab (Section Properties Dialog Box) (page 2272)

    ---

# Editing Sample Line Group Properties

You can specify sample line group properties, and by extension the sample lines in that group, that control the group's content and format.

For example, for an existing sample line group, you can use the Sample Line Group Properties dialog box (page 2257) tabs to apply major changes to a sample line group by changing general properties:

■ **Information**. Specifies the sample line group's name and description.

■ **Sample Lines**. Specifies the default properties of the sample lines that are included in the sample line group. Also, you can specify the default label style. By changing the default label style here, you instantly change the sample line's displayed appearance.

■ **Sections**. Specifies the properties that draw and manage the section associated with the current sample line group.

■ **Section Views**. Specifies the properties that draw and manage the section view associated with the current sample line group.

■ **Material List**. Specifies the properties of material lists associated with the current sample line group. Each list represents quantity takeoff calculation criteria for the specified materials.

Sample line properties display and specify data only about that sample line. Sample line group properties display and specify rules for creating sample lines by range of stations or from corridor stations.

For more information, see Editing Sample Line Properties (page 1149).

**To edit sample line group properties**

1 In Toolspace, on the Prospector tab, right-click the sample line group name you want to edit. Click Properties.

2 In the Sample Line Group Properties dialog box (page 2257), review or change properties on the following tabs: Information, Sample Lines, Sections, Section Views, and Material List.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <site name> ➤ Alignments ➤ <alignment name> ➤ Sample Line Group ➤ <sample line group name> ➤ Properties

Command Line

EditSampleLineGroupProperties

Dialog Box

Sample Line Group Properties (page 2257)

# Editing Sample Line Properties

When you create a sample line you can specify properties to control the sample line's content and format.

For example, for an existing sample line group, you can use the Sample Line Properties dialog box (page 2261) tabs to apply major changes to a sample line by changing general properties:

■ **Information**. Specifies the sample line's name, text description, and object style. By changing the style here, you instantly change the sample line's displayed appearance.

- **Sample Line Data**. Specifies the sample line's group name, associated alignment, station information, and whether the sample line locks to the station. Also, you can specify the label style. By changing the label style here, you instantly change the sample line's displayed appearance.

- **Sections**. Specifies the properties that draw and manage the section associated with the current sample line. For any sample line, you can also change its style.

- **Section Views**. Specifies the properties that determine which section views are created from the current sample line.

---

**NOTE** Sample line properties display and specify data only about that sample line. Sample line group properties display and specify rules for creating sample lines by range of stations or from corridor stations.

---

**To edit sample line properties**

1 In the drawing, click the sample line you want to edit.

2 Right-click the sample line. Click Sample Line Properties.

3 Review or change properties on the tabs of the Sample Line Properties dialog box.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <site name> ➤ Alignments ➤ <alignment name> ➤ Sample Lines ➤ <sample line name> ➤ Properties

Object Shortcut Menu

Right-click <sample line object> (in the drawing) ➤ Sample Line Properties

Command Line

EditSampleLineProperties

Dialog Box

Sample Line Properties (page 2261)

## Editing Section Properties

When you create a section you can specify properties to set the section's name, type, object style, station details, and labeling.

Open the Section Properties dialog box (page 2272) for an existing section to verify the name of the section and review other information.

Apply changes to a section by changing its general properties.

The **Information** tab specifies the section's name, description, and object style. By changing the style here, you instantly change the section's displayed appearance.

The **Section Data** tab specifies and displays tabular data with a single row listing the properties of the section. Some of these properties, such as layer, mode, and style can be edited. The other properties are only for review.

**To edit section properties**

1 In the drawing, click the section line you want to edit.

**2** Right-click the section line. Click Section Properties.

**3** Review or change properties on the tabs of the Section Properties dialog box (page 2272).

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <site-name> ➤ Alignments ➤ <alignment name> ➤ Sample Line Groups ➤ <sample line group name> ➤ Sections ➤ <section name> ➤ Properties

Object Shortcut Menu

Right-click <section object> (in the drawing section view) ➤ Section Properties

Command Line

SectionProperties

Dialog Box

Section Properties (page 2272)

# Editing Corridor Section Properties

When you create a corridor section you can specify properties to set the corridor section's name, type, object style, layer, and codes.

Open the Corridor Section Properties dialog box (page 2229) for an existing corridor section to verify the name of the corridor section and review other information. Apply changes to a corridor section by changing its general properties:

■ **Information**. Specifies the corridor section's name, text description, and object style. By changing the style here, you instantly change the corridor section's displayed appearance.

■ **Section Data**. Specifies and displays tabular data with a single row listing the properties of that corridor section. Some of these properties, such as layer, mode, and style can be edited. The other properties are only for review.

■ **Codes**. Displays codes and style information for review. For information about changing the display style of various point, link, and shape codes, see Using Codes and Code Set Styles (page 1405).

**To edit corridor section properties**

**1** On the prospector, click the corridor section collection that contains the corridor section you want to edit.

**2** On the item view, right-click the individual corridor section  . Click Properties.

**3** Review or change properties on the tabs of the Corridor Section Properties dialog box (page 2229).

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Alignments ➤ <alignment-type> ➤ <alignment name> ➤ Sample Line Groups ➤ <sample line group name> ➤ Sections ➤ <corridor section group name>. In the item view, right-click <corridor section name>, click Properties

Command Line

SectionCorridorProperties

Dialog Box

Corridor Section Properties (page 2229)

# Editing Pipe Network Section Properties

When you create a pipe network section you can specify properties to set the pipe network section's name and description and view its object style, layer, and data source.

Open the Pipe Network Section Properties dialog box (page 2253) for an existing pipe network section to verify the name of the section and review other information.

**To edit and view pipe network section properties**

1  On the prospector, click the pipe network section collection that contains the pipe network section you want to edit.

2  On the item view, right-click the individual pipe network section ⬚. Click Properties.

3  Review or change properties on the tabs of the Pipe Network Section Properties dialog box (page 2253).

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Sites ➤ <site-name> ➤ Alignments ➤ <alignment name> ➤ Sample Line Groups ➤ <sample line group name> ➤ Pipe Network Sections ➤ <pipe network section group name>. In the item view, right-click <pipe network section name>, click Properties

Dialog Box

Pipe Network Properties (page 2253)

# Creating and Editing Section Views

AutoCAD Civil enables you can to easily create, manage, and edit section views.

Section views are created from existing sample lines and sections. For more information, see Creating and Editing Sample Lines and Sections (page 1142).

A section view consists of a grid on which one or more sections are displayed as graphed lines. Multiple section views can be plotted on a sheet that is a specified size and configuration.

A section view is very similar to a profile view. It consists of a grid or graph with specific characteristics that are controlled by section view styles. Bands can also be displayed above or below the section view.

A section view can display one or more of the available sections at the sample line. To plot multiple sections, use a paperspace viewport size to set up the sheet.

Configure bands and section annotations in a section view to make the section view clearer or more informative for the user.

# Creating/Editing Section Views

Create a section view to display an existing ground section, using sample lines cut across a horizontal alignment.

Sample lines (1) must be created prior to creating a section view (2). Section views can display existing ground surface sections, corridor sections, and pipe sections.
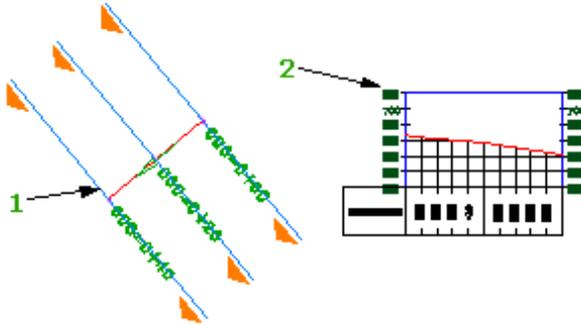


Each section view must be linked to an alignment. The section view displays new or existing sections. Sections can also be created without displaying them on a section view. When you create a section view for an alignment, the complete list of sections for the alignment is displayed. Choose which sections to include in the section view.

Create either an EG (existing ground) section view or multiple section views. You can change the display style of a section view, and add labels and bands.

**To create a section view**

1 Click Home tab ➤ Profile & Section Views panel ➤ Section Views drop-down ➤ Create Section View . 

2 In the Create Section View wizard (page 2239), click **Create Section View** to accept location, style, offset, elevation, display, data band, and volume table defaults. You can later change the default settings in the Section View Properties dialog box (page 2281). Alternatively, you can change settings in the following pages of the wizard

- General page (page 2146) – Change the location, name, layer and style of the new section view.

- Offset Range page (page 2146) – Change the width of the offsets of the new section view.

- Elevation Range page (page 2146) – Change the maximum and minimum elevation of the new section view.

- Section Display Options page (page 2148) – Change the visibility and style of the new section view.

- Data Bands page (page 2150) – Change the band set and data band location for the new section view.

- Section View Tables page (page 2152) – Change the type and style of volume tables, and their position relative to, the new section view.

**NOTE** A material list must exist in the drawing, before you can modify **Section View Tables** settings.

3 In the drawing, find a suitable location for the section view, for example, outside the surface boundary for ease of viewing. Click that location to define the origin.

The section view is displayed.

**To create multiple section views**

1   Click Home tab ➤ Profile & Section Views panel ➤ Section Views drop-down ➤ Create Multiple Views
    ![icon] .

2   In the Create Multiple Section Views wizard (page 2232), click **Create Section Views** to accept location, style, offset, elevation, display, data band, and volume table defaults. You can later change the default settings through the Section View Group Properties (page 2279) dialog box. Alternatively, you can change settings in the following pages of the wizard:

    ■ General page (page 2233) – Change the location, name, layer and style of the new section views.

    ■ Offset Range page (page 2233) – Change the width of the offsets of the new section views.

    ■ Elevation Range page (page 2234) – Change the maximum and minimum elevations of, and group options for, the new section views.

    ■ Section Display Options page (page 2234) – Change the visibility and style of the new section views.

    ■ Data Bands page (page 2234) – Change the band set and data band location for the new section views.

    ■ Section View Tables page (page 2235) – Change the type and style of volume tables, and their position relative to, the new section views.

    ---

    **NOTE** A material list must exist in the drawing, before you can modify **Section View Tables** settings.

    ---

3   In the drawing, find a suitable location for the multiple section views, for example, well outside the surface boundary for ease of viewing. Click that location to define the origin.

    The multiple section views are displayed.

You can plot an entire sample line group, or you can specify a user-defined station range (1) to control how many section views (2) are created.
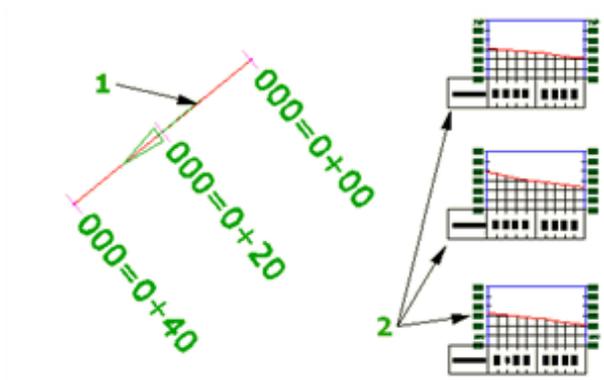


**To change the display style of a section**

1   In the section view, click the section line you want to change.

2   Right-click the section line. Click Section Properties.

3   In the Section Properties dialog box, click the Information tab (page 2272).

4   In the Object Style field, select a different style in the list.

**To change the display style of a section view**

1   In the section view, click the section view grid

**2** Right-click the grid. Click Section View Properties.

**3** In the Section View Properties dialog box, click the Information tab (page 2281).

**4** In the Object Style field, select a different style in the list.

**To add a section view label**

**1** Zoom to the section view area to clearly see the section view and its annotations.

**2** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Section View ➤ Add Section View Labels  .

**3** In the Add Labels dialog box (page 1970), for Label Type, select the type of label (either Grade or Offset Elevation) in the list.

**4** Optionally, for style, select the style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

**5** Click Add.
On the command line, you are prompted to select a section view.

**6** In the drawing, click a graph line of the section view you want to label.
On the command line, you are prompted to pick a point.

**7** In the section view, click the point, or points for a grade label, that define the label.
The label is displayed at the desired location in the section view.

**8** Right-click or press Esc to end the session.

**9** In the Add Labels dialog box, click Close.

**To add bands to a section view**

**1** In the section view, click the section view grid.

**2** Right-click the grid. Click Section View Properties.

**3** In the Section View Properties dialog box, click the Bands tab (page 2283).

**4** In the List Of Bands area, select either Bottom or Top as the location for the new data band in the section view.

**5** Select a band type and style. Click Add.

**6** To add more bands to the section view, repeat Steps 4 and 5.

**7** Optionally, on the right side of the List Of Bands, move selected bands up  or down  in the stack.
To delete any unwanted bands, click  .

**8** To save this arrangement of bands as a set for reuse, click Save As Band Set.

**9** Click OK to close the Section View Properties dialog box. The bands are displayed in the section view.

**To add profile grade points to a section view**

**1** In the section view, click the section view grid.

**2** Right-click the grid. Click Section View Properties.

**3** In the Section View Properties dialog box, click the Profile Grade Lines tab (page 2287).

**4** In the Alignment area, click an alignment in the list or click  to pick an alignment from the drawing.

**5** Click Add.

**6** To add more grade point labels to the section view, repeat Steps 4 and 5.

**7** Optionally, to delete any unwanted grade points, select them in the list. Click  .

**8** Click OK to close the Section View Properties dialog box. The profile grade points are displayed in the section view.

# Editing Section View Properties

When you create a section view you can specify properties to control a section view's content and format.

In the Section View Properties dialog box, edit offset range, elevation range, section data, band information, projected object information, volume table information and profile grade line information.

Apply changes to a section view by changing its general properties:

- **Information**. Specifies the section view's name, description, and object style. By changing the style here, you instantly change the section's displayed appearance.

- **Offsets**. Displays the properties of the selected section view and the section view grid on which sections are displayed.

- **Elevation**. Specifies the elevation properties of the selected section view and the section view grid on which sections are displayed.

- **Sections**. Specifies which sections to display in the section view. For any section, you can also override its style and label set to display it differently in the current section view.

- **Bands**. Specifies which bands to include in the section view, and whether to place them along the top or bottom of the grid. Data is displayed for specified section objects.

- **Volume Tables**. Specifies table type and style, and the associated material, material list and layer.

- **Profile Grade Lines**. Specifies a profile and displays its elevation by marking it at the appropriate offset and elevation in the section view.

- **Projection**. Specifies the name, style and elevation of projected objects in a section view.

For more information, see Section View Group Properties Dialog Box (page 2279).

**To edit section view properties**

**1** In the drawing, click any grid or axis line of the section view you want to edit.

**2** Right-click the section view grid. Click Section View Properties.

**3** Review or change properties on the tabs of the Section View Properties dialog box (page 2279).

**To edit section view group properties**

**1** In the drawing, click any grid or axis line of any section view in the group you want to edit.

**2** Right-click the section view grid. Click Section View Group Properties.

**3** Review or change properties on the tabs of the Section View Group Properties (page 2279) dialog box.

### Quick Reference

# Adding Projected Objects to a Section View

Project AutoCAD or AutoCAD Civil objects from plan view into a section view.

You can project AutoCAD points, blocks, 3D solids, and 3D polylines; you can also project COGO points, feature lines, and survey figures.

For more information, see Working with Projected Objects (page 1118).

**To project objects to a section view**

1 Home tab ➤ Profile & Section Views panel ➤ Section Views drop-down ➤ Project Objects To Section View  .

2 Click one or more objects in the drawing that you want to project. Right-click or press Enter after all objects are selected.

3 Click the section view in which you want the objects to appear.

### Quick Reference

Ribbon

Home tab ➤ Profile & Section Views panel ➤ Section Views drop-down ➤ Project Objects To Section View 

Command Line

ProjectObjectsToSect

# Editing Projected Objects in a Section View

Use the shortcut menu for a projected object for quick access to object properties.

When you right-click a projected object in a section view, the shortcut menu is displayed. The only menu selection specific to the projection is Projection Object Properties. It opens the Projections tab (page 2287) of the Section View Properties dialog box.

For more information, see Working with Projected Objects (page 1118).

# Removing a Projected Object from a Section View

Select a projected object and remove it from the section view.

Other projected objects not selected in the section view remain in place.

For more information, see Working with Projected Objects (page 1118).

**To remove a projected object**

1   In the section view, click the object to want to remove.

2   In the Modify Projected Object panel, click Remove From View  .

## Quick Reference

Ribbon

Projected Object tab ➤ Modify Projected Object panel ➤ Remove From View 

Command Line

RemoveFromSectionView

# Sections Command Reference

The AutoCAD Civil commands for section-related functionality and a brief description of their functionality.

| Command | Description |
| --- | --- |
| AddSectionViewLabels | Adds a grade label after you select an existing section view and two points to define grade start and finish (page 1139) |
| AddSectionViewGradeLbl | Adds a grade label after you select an existing section view and two points to define grade start and finish (page 1139) |
| AddSectionViewOffElevLbl | Adds an offset elevation label after you select an existing section view and desired point to define the label location (page 1139) |
| AddSheet | Adds a sheet, on which to draw section views, to the drawing after you select an origin (page 1153) |
| CreateCrossSectionStyle | Creates a new section style (page 1132) |

| Command | Description |
|---|---|
| CreateGroupPlotStyle | Creates a new group plot style (page 1132) |
| CreateMultipleSectionView | Creates multiple section views (page 1153) |
| CreateSampleLines | Creates sample lines based on existing alignments (page 1142) |
| CreateSampleLineStyle | Creates a new sample line style (page 1131) |
| CreateSectionView | Creates a new section view (page 1153) |
| CreateSectionViewStyle | Creates a new section view style (page 1133) |
| EditCrossSection | Edits a selected section (page 1148) |
| EditCrossSectionStyle | Edits the section style after you select a section line in a section view graph (page 1133) |
| EditGroupPlotStyle | Edits a group plot style (page 1132) |
| EditSampleLine | Edits a sample line after you select an alignment and the sample line (page 1145) |
| EditSampleLineLabels | Edits the labels of a selected sample line (page 1137) |
| EditSampleLineGroupProperties | Edits sample line group properties after you select a sample line (page 1149) |
| EditSampleLineProperties | Edits sample line properties after you select a sample line (page 1149) |
| EditSampleLineStyle | Edits an existing sample line style (page 1131) |
| EditSectionLabels | Edits the labels of a selected section (page 1034) |
| EditSectionViewStyle | Edits an existing section view style (page 1133) |
| EditSheetProperties | Edits the sheet properties after you select a sheet (page 2295) |
| EditSheetStyle | Edits the sheet style after you select a sheet. (page 1133) |
| MakeSampleLineOrthogonal | Returns sample line(s) to the perpendicular from their parent alignment (page 1146) |
| SectionProperties | Edits section properties after you select a section line in a section view graph (page 1150) |
| ProjectObjectToSect | Projects an object into a section view (page 1157) |
| RemoveFromSectionView | Removes a projected object from a section view (page 1158) |

# Material and Quantity Analysis

# 26

Use materials and sectional or profile information to extract and report material volumes.

◯ Material and Quantity Analysis Workflow

## Understanding Material and Quantity Analysis

In AutoCAD Civil, you can create, manage, and plot quantities.

You then save or render them using:

- Total volume tables
- Material volume tables
- Volume reports
- Mass haul diagrams

Creating material lists (page 1163) is the essential step in the creation of volume tables and reports. Creation of tables and reports can be performed once the material lists are generated. As part of the production of the material lists, you will need to set quantity takeoff criteria (page 1165).

As with the creation of volume tables and reports, the creation of material lists is also a prerequisite to creating mass haul diagrams (page 1174).

## Quantity Takeoff Settings

Use quantity takeoff settings to specify the default style and name format settings for quantity takeoff and the default behavior for quantity takeoff-related commands.

Settings are handled in a standard way throughout AutoCAD Civil. Access settings using the Toolspace Settings tree. Control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

Use the Toolspace Settings tree Quantity Takeoff collection shortcut menu to establish defaults for all of the quantity takeoff-related features and commands. You can change quantity takeoff-specific settings at this level, or you can override the drawing ambient settings.

Use the Commands collection under the ⊞ Quantity Takeoff collection to change quantity takeoff settings for a specific command.

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**NOTE** Overrides to the drawing ambient settings at the Quantity Takeoff collection level and the quantity takeoff Commands collection level affect only the specified level. The drawing level settings are not changed.

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The topics in this section describe only those settings that affect quantity takeoff-related features and commands. They do not cover the drawing ambient settings that you can change at the quantity takeoff collection level and the quantity takeoff command level, even though those settings are displayed in the Edit Feature Settings - Quantity Takeoff dialog box. For more information about the drawing ambient settings, see Specifying Ambient Settings (page 68).

## Editing Quantity Takeoff Settings

Change quantity-takeoff-related settings before you create criteria, material lists, tables, or reports.

For example, before creating a quantity takeoff table, you may want to specify the default table format.

If a closed lock appears in the Lock column for a property, the property is locked at a higher level of the Settings tree. It cannot be changed at this level.

If you change a property value, a check mark is placed in the Override column. This override is also noted in the property table for related objects higher up in the Toolspace Settings tree, where an arrow appears in the Child Override column.

**To change quantity takeoff settings**

1 Do one of the following:
   - **To edit the general quantity takeoff settings**: In Toolspace, on the Settings tab, right-click the Quantity Takeoff collection and click Edit Feature Settings.
   - **To edit the settings for a specific command**: In Toolspace, on the Settings tab, expand the Commands collection in the Quantity Takeoff collection. Right-click the command for which you want to change settings.

   The Edit Feature Settings - Quantity Takeoff dialog box (page 2305) is displayed.

2 To specify the default quantity takeoff criteria and material shape style, expand the ⊞ Default Styles property group, and edit the Quantity Takeoff Criteria and Material Shape Style settings.

3 To specify the default naming for quantity takeoff materials and material lists, expand the ⊞ Default Name Format property group, and edit the Material Name Template and Material List Name Template settings.

4 To modify the default material volume table settings, expand the Table Creation property group, and edit the standard table creation options. This property group is available if you accessed the settings from the AddMaterialVolumeTable command in Step 1.

5 To modify the default total volume table settings, expand the Table Creation property group, and edit the standard table creation options. This property group is available when you access the settings from the AddTotalVolumeTable command in Step 1.

6 To modify the default material definition settings, expand the Define Materials Options property group, and edit the curve correction settings. For more information, see Generating Material Lists (page 1163). This property group is available when you access the settings from the ComputeMaterials command in Step 1.

**7** To modify the default quantity takeoff report settings, expand the Quantity Report Options property group, and edit the XML report display setting. This property group is available when you access the settings from the GenerateQuantitiesReport command in Step 1.

**8** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: right-click Quantity Takeoff collection ➤ Edit Feature Settings
OR
Settings tab: Quantity Takeoff ➤ Commands ➤ right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Quantity Takeoff (page 2305)

# Analyzing Material Volumes

Use the material volumes functionality to extract and report sectional material volumes, based on sample line groups.

# Material Lists and Material Analysis

Create material lists and quantity takeoff criteria as the first step in analyzing material volumes.

You can create tables and reports for:

■ Volumes along an alignment, comparing various design surfaces and existing ground surfaces.

■ Volumes for shapes, which are closed cross-sectional areas created by a single subassembly. For example, a curb (a closed area within a concrete curb or curb and gutter).

■ Volumes between various (designed/grading/existing) formations.

---

**TIP** To query composite and bounded volume differences between surfaces, use surface volume reporting. For information, see Calculating Surface Volumes (page 690).

---

Material volume reporting uses criteria settings that are portable and extensible. You can create criteria based either on existing data, including surfaces and sample line groups, or on standard surface names.

You prepare to generate material volume information by creating a list of materials and applying the predefined criteria to it, mapping existing surfaces or other objects to the names in the criteria. After the material list is generated, the settings and volume calculations are stored with the sample line group and can be used to generate tables and reports.

You can display material volume information using standard AutoCAD Civil table formats, or view and export the information in an XML format file.

## Generating Material Lists

Use the Compute Materials command to generate a material list for a sample line group.

Material lists are the starting points for creating quantity takeoff tables and reports, and mass haul diagrams. Each time you apply a criteria to a sample line group, a material list is added to the sample line group properties. You can then use the material list to create a volume table or report, or a mass haul diagram.

Before you create a material list, you must first define criteria and create sample lines for the alignment along which you are going to generate quantity takeoff or mass haul information.

The material lists you create are stored as part of the sample line group properties. If you select a sample line group that already has a material list when you are using the Compute Materials command, the existing list is opened for editing.

**Curve Correction**

Use curve correction when a curve correction tolerance value is exceeded. Curve correction may be required to correct volume calculations within a curvilinear section of an alignment (or on two sides of a horizontal intersection point). If the sample lines are within a specified tolerance or if the angle of deflection between the successive stations is small, then curve correction is not used.

**To create a new material list for a sample line group**

1  Click Analyze tab ➤ Volumes And Materials panel ➤ Compute Materials 🖥 .

2  In the Select a Sample Line Group dialog box, click the Select Alignment field. Select an alignment from the list or click 🔲 and select an alignment in the drawing.

3  Click the Select Sample Line Group field. Select a sample line group or click 🔲 to select a group in the drawing.

4  Click OK.

   If there is no material list associated with the sample line group, the Compute Materials dialog box (page 2300) is displayed.

   If one or more material lists are already part of the sample line group properties, the Edit Material List dialog box (page 2307) is displayed. You can add a new materials list or edit an existing list as required.

5  Click the Quantity Takeoff Criteria field. Select the criteria.

6  Optionally, select the Curve Correction Tolerance check box and enter a curve correction value.

7  Optionally, if the criteria uses the same surface and structure names as objects in the drawing, you can quickly map the names by clicking Map Objects With Same Name.

8  Click OK.

   The quantity takeoff calculation is performed and the material list is added to the sample line group properties.

## Quick Reference

Ribbon

Analyze tab ➤ Volumes And Materials panel ➤ Compute Materials 🖥

Menu

Sections menu ➤ Compute Materials

Command Line

ComputeMaterials

Dialog Box

Compute Materials (page 2300)

## Creating Quantity Takeoff Criteria

Create quantity takeoff criteria in preparation for generating quantity takeoff tables or reports.

The criteria are stored in the Toolspace Settings tab under the Quantity Takeoff collection. After you create a criteria, you can apply it to a sample line group to create a material list. You use the material list to generate quantity takeoff tables and reports.

To create a criteria, you create a list of materials and shapes. For example, you can create an entry that calculates the volume of material to remove by creating a material called Ground Removed, and then add to it two surfaces that are compared to generate the volume information.

You can also define materials in a criteria using a sample line group. Select a sample line group, and then create material definitions using the list of surfaces that are used as a data source for the group.

### Quantity Types

Quantity takeoff supports the following quantity types:

- **Cut**. Used to calculate the material to remove. For example, this could be the amount below an existing ground (EG) surface and above a finished ground (FG) surface:
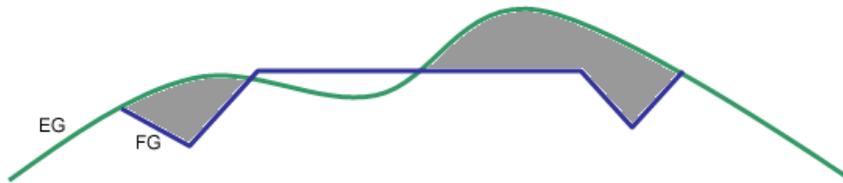


- **Fill**. Used to calculate the material to add. For example, this could be the amount above an existing ground (EG) surface and below a finished ground (FG) surface:
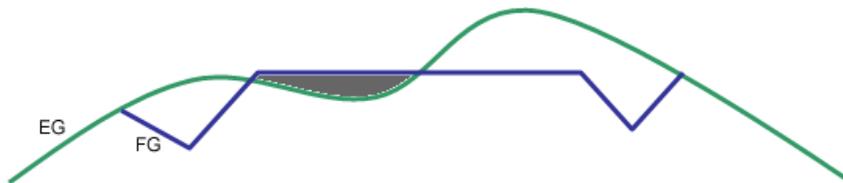


- **Cut and Refill**. Used to calculate the material to remove and add based on a refill factor that is different from the cut factor. For example, an existing subsurface material area for a corridor may have to be removed if it comprises loose soil or marsh land, and refilled with a different type of material to provide structural stability.

- **Earthworks**. Used to calculate the total cut and fill volumes. For example, this could be any differences between the existing ground (EG) surface and the finished ground (FG) surface:
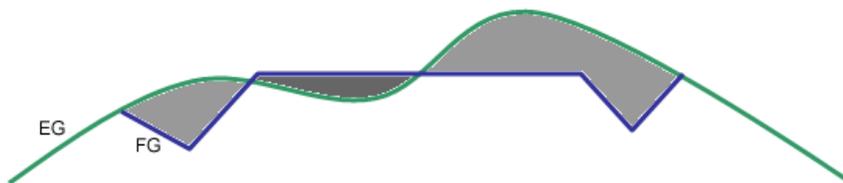


- **Structures**. Used to calculate the volumes of corridor shapes, for example, in the following illustration, which is a cross-section of a corridor, the volume of the sidewalk shape would be calculated:
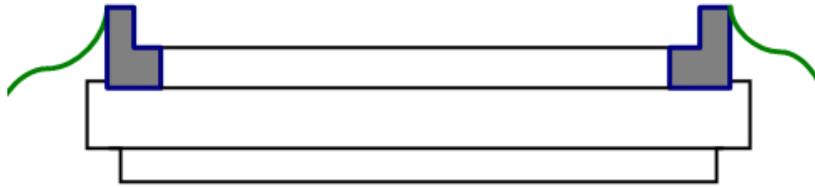
**Cut, Fill, and Refill Factors**

Use Cut, Fill, and Refill factors to adjust volumes to be hauled. As the hauled volumes are used as a baseline set at an even factor of 1.0. The adjustment factors are:

■ **Cut factor** - As the volume of material generally expands after it is removed, the cut factor is usually set to greater than 1.0, indicating swell or expansion. For example, a 1.2 cut factor would mean that for every 1.0 cubic meter of material removed, 1.2 cubic meters of volume would need to be accounted for transport.

■ **Fill factor** - As the material generally compacts when used as fill, the fill factor is usually set to less than 1.0, which indicates compaction or shrinkage of the material. For example, a 0.8 fill factor would mean that for every 0.8 cubic meter of material required for the fill site, 1.0 cubic meter of that material would need to be transported.

■ **Refill factor** - As this factor depends on the cut material type and other considerations, it can run across a range. A heavy aggregate might have a Refill factor of 1.0, while fill cut from a rock ledge might have a Refill factor of 1.2. Not all cut material may even be reusable, as when it is cut from bog or marsh. The refill factor would then be 0.

**Creating new criteria**

1  In Toolspace, on the Settings Tree, expand the  Quantity Takeoff collection. Right-click the Quantity Takeoff Criteria collection. Click New.

2  In the Quantity Takeoff Criteria dialog box, click the Information tab (page 2321).

3  To edit the name of the criterion, enter a new name in the Name field.

4  To edit the description of the criterion, enter a new description in the Description field.

5  Click Apply to make the changes, or click OK to make the changes and close the Quantity Takeoff Criteria dialog box.

6  To define or edit the criterion, click the Material List tab (page 2321).

7  Click Add New Material.

8  Select the new material. Click the Quantity Type field. Select the quantity type, either Cut, Fill, Cut and Refill, Earthworks, or Structures.

9  Click the Shape Style field. Select the default style used to display the material section in a section view.

10 Optionally, edit the cut, fill, and refill factors by selecting the corresponding fields. Enter new values.

11 If you selected a quantity type of Cut, Fill, Cut and Refill, or Earthworks to add surfaces, select the new material. From the Data Type list, select Surface. In the Select Surface list, enter the surface name or select a surface.

12 If you selected a quantity type of Structures, to add corridor shapes, select the new material. Select Corridor Shapes from the Data Type list. In the Select Shape list, enter the shape name or select a shape. Click the Shape Style field. Select a style for the shapes.

**13** For each data type added to a material, click the Condition field. Select the condition.

**14** Optionally, repeat Steps 7 through 13 to create all required materials and surfaces.

**15** Click Apply.

**Defining criteria from a sample line group**

**1** In Toolspace, on the Settings Tree, expand the  Quantity Takeoff collection. Right-click the Quantity Takeoff Criteria folder and click New.

**2** In the Quantity Takeoff Criteria dialog box, click the Information tab (page 2321).

**3** To edit the name of the criteria, enter a new name in the Name field.

**4** To edit the description of the criteria, enter a new description in the Description field.

**5** Click Apply to make the changes, or click OK to make the changes and close the Quantity Takeoff Criteria dialog box.

**6** To define the criteria, click the Material List tab (page 2321).

**7** Click Define From A Sample Line Group

**8** In the Define Material Criteria dialog box, click the Select Alignment field. Select the alignment or click  to select an alignment in the drawing.

**9** Click the Select Sample Line Group field to select a sample line group.

**10** Select the surfaces to use in the criteria by selecting or clearing the check box in the Select field next to the surface name.

**11** Click OK.

**12** In the Quantity Takeoff Criteria dialog box, click Apply or OK.

## Quick Reference

Toolspace Menu

Settings tab: Quantity Takeoff ➤ Quantity Takeoff Criteria ➤ New

Dialog Box

Quantity Takeoff Criteria (page 2321)

# Analyzing Sectional Volumes

Use the quantity takeoff functionality to extract and report sectional material volumes, based on sample line groups.

You can create tables and reports about:

■ Volumes along an alignment, comparing various design surfaces and existing ground surfaces.

■ Volumes for shapes, which are closed cross-sectional areas created by a single subassembly. For example, a curb (a closed area within a concrete curb or curb and gutter).

■ Volumes between various (designed/grading/existing) formations.

**TIP** To query composite and bounded volume differences between surfaces, use surface volume reporting. For information, see Calculating Surface Volumes (page 690).

Quantity takeoff reporting uses criteria settings that are portable and extensible. You can create criteria based either on existing data including surfaces and sample line groups or on standard surface names.

You prepare to generate quantity takeoff information by creating a list of materials and applying the predefined criteria to it, mapping existing surfaces or other objects to the names in the criteria. After the material list is generated, the settings and volume calculations are stored with the sample line group and can be used to generate tables and reports.

You can display quantity takeoff information using standard AutoCAD Civil table formats or view and export it in an XML format file.

## Analyzing Sectional Volumes

Use the table and report creation commands to analyze sectional volumes.

The results of a quantity takeoff calculation are displayed using one of the following three methods:

- Total Volume Table

- Material Volume Table

- Quantities Report

A Total Volume Table contains cut, fill, and cumulative volume information (for example, an earthworks or cut/fill report).

A Material Volume Table contains cut, fill, and cumulative volume information for a specific material in the material list (for example, the cumulative volume of a structure element).

A Quantities Report is an XML format file that contains the criteria definition (comparable surfaces), material types (such as cut type), shrink and swell factors, as well as refill factors. The specific format is determined by the style sheet you select.

Before you create a quantity takeoff report or table, materials must be defined in the Sample Line Group properties or computed using the Compute Materials command.

**Style Sheets**

Three style sheets installed with AutoCAD Civil are used to create external quantity takeoff reports:

- *Earthworks.xsl*. Reports station-by-station values in a tabular format for cut and fill volumes, incremental volumes, and cumulative net volume.

- *Select Material.xsl*. Reports station-by-station values for the selected materials. At each station, all selected materials defined in this criteria and cumulative volumes are reported.

- *Mass Haul - Multiple Materials.xsl*. If you have defined many material types to remove (for example, for embankment creation for road widening along the corridor), you can use this style sheet to aggregate the material types and produce material-by-material reports at each station as well as aggregate volume reports for mass hauls.

**To create a total volume table**

1 Click Analyze tab ➤ Volumes And Materials panel ➤ Total Volume Table 📊 .

2 In the Create Total Volume Table dialog box (page 2303), select a table style and layer.

**3** Click the Select Alignment field. Select an alignment from the list or click  to select an alignment in the drawing.

**4** Click the Select Sample Line Group field. Select a sample line group from the list.

**5** Click the Select Material List field. Select a material list from the list.

**6** Optionally, select Split Table, and then specify the maximum and minimum number of rows in a table, the distance to offset tables from each other, and the method for arranging multiple tables.

**7** Under Behavior, specify whether the table is static or dynamic.

**8** Click OK.

The upper-left corner of the new table is attached to your cursor.

**9** Click in the drawing to set the location of the table.

**To create a material volume table**

**1** Click Analyze tab ➤ Volumes And Materials panel ➤ Material Volume Table  .

**2** In the Create Material Volume Table dialog box (page 2303), select a table style and layer.

**3** Click the Select Alignment field. Select an alignment from the list or click  to select an alignment in the drawing.

**4** Click the Select Sample Line Group field. Select a sample line group from the list.

**5** Click the Select Material List field. Select a material list from the list.

**6** Click the Select a Material field. Select a material from the list.

**7** Optionally, select Split Table, and then specify the maximum and minimum number of rows in a table, the distance to offset tables from each other, and the method for arranging multiple tables.

**8** Under Behavior, specify whether the table is static or dynamic.

**9** Click OK.

The upper-left corner of the new table is attached to your cursor.

**10** Click in the drawing to set the location of the table.

**To generate an external quantity takeoff report**

**1** Click Analyze tab ➤ Volumes And Materials panel ➤ Volume Report 

**2** In the Report Quantities dialog box (page 2323), click the Select Alignment field. Select an alignment from the list or click  to select an alignment in the drawing.

**3** Click the Select Sample Line Group field. Select a sample line group from the list.

**4** Click OK.

**5** Click the Select Material List field. Select a material list from the list.

**6** Enter a style sheet name in the Select A Style Sheet field or click  to browse for one.

**7** To display the quantity takeoff report, select the Display XML Report check box.

**8** Click OK to save the report settings and generate the report.

### Quick Reference

Ribbon

Analyze tab ➤ Volumes And Materials panel ➤ Total Volume Table 

Analyze tab ➤ Volumes And Materials panel ➤ Material Volume Table 

Analyze tab ➤ Volumes And Materials panel ➤ Volume Report 

Menu

Sections menu ➤ Add Tables ➤ Total Volume
Sections menu ➤ Add Tables ➤ Material Volume
Sections menu ➤ Generate Volume Report

Command Line

AddTotalVolumeTable
AddMaterialVolumeTable
GenerateQuantitiesReport

Dialog Box

Total Volume Table Creation (page 2303)
Material Volume Table Creation (page 2303)
Report Quantities (page 2300)

## Quantity Takeoff Table Styles

Styles control the way quantity takeoff tables are displayed in a drawing.

In AutoCAD Civil, all objects have a standard object style grouping on the Settings tree, called an object style collection. You use this to create, edit, copy, and delete the styles for that object. For more information, see The Object Style Collection (Settings Tree) (page 94).

## Creating and Editing Quantity Takeoff Table Styles

Use the Toolspace Settings tree to copy, edit, or delete a quantity takeoff table style.

Two types of quantity takeoff table styles are listed in the Settings tree under the Quantity Takeoff collection: Total Volume and Material. The two types of table styles have the same properties as other AutoCAD Civil table styles except for the property fields that can be included in the table.

If you generate a quantity takeoff report, the style of the external file is controlled using the specified style sheet. For more information, see the report generation information in Analyzing Sectional Volumes (page 1168).

Create a new quantity takeoff table style by copying an existing style. Then change its properties to suit your requirements. Use the Table Style dialog box (page 2437) tabs to specify the main properties.

---

**NOTE** Before editing an existing quantity takeoff table style to change its properties, remember that any changes you make will apply to tables that use the style. If you want to change the style of some quantity takeoff tables but not all, consider creating a new style.

---

**To create a new quantity takeoff table style**

1   In Toolspace, in the Settings tree, under Quantity Takeoff, expand the Table Styles collection.

**2** For the Total Volume or Material collection, do one of the following:

- Right-click the collection. Click New.

- Right-click the existing style to use as a template. Click Copy.

**3** In the Table Style dialog box (page 2437), enter or change the name of the style and other settings as required.

**4** Click OK.

**To copy an existing quantity takeoff table style**

**1** In Toolspace, in the Settings tree, under Quantity Takeoff, expand the Table Styles collection.

**2** Expand the Total Volume or Material collection.

**3** Right-click an existing style. Click Copy.

**4** In the Table Style dialog box (page 2437), edit the name of the style and other settings as required.

**5** Click OK.

**To edit a quantity takeoff table style**

**1** In Toolspace, in the Settings tree, under Quantity Takeoff, expand the Table Styles collection.

**2** Expand the Total Volume or Material collection.

**3** Right-click the style you want to edit. Click Edit.

**4** In the Table Style dialog box (page 2437), change settings as required.

**5** Click OK.

**To delete a quantity takeoff table style**

**1** In Toolspace, in the Settings tree, under Quantity Takeoff, expand the Table Styles collection.

**2** Expand the Total Volume or Material collection.

**3** Right-click the style to delete. Click Delete.

**4** In the confirmation dialog box, click Yes.

## Quick Reference

Toolspace Shortcut Menu

Create: Settings tab: right-click Total Volume or Material collection ➤ New
Copy: Settings tab: right-click Total Volume or Material collection ➤ Copy
Edit: Settings tab: right-click Total Volume or Material collection ➤ Edit
Delete: Settings tab: right-click Total Volume or Material collection ➤ Delete

Dialog Box

Table Style (page 2437)

# Using Mass Haul Diagrams

Use mass haul diagrams to display earthwork volumes along an alignment, based on sectional volume computation.

The use of mass haul (page 2477) diagrams helps designers and contractors understand where gross material movements occur and as key indicators used to compare the economies of alternative designs. These diagrams are typically presented along with profile views for design review. They are used to analyze the following aspects of the design:

■ Distance over which the cut and fill will balance

■ Amount of material to be moved and the direction of movement

■ Identification of borrow pit (page 2469)s and dump site (page 2473)s

To generate mass haul diagrams, you need an alignment, a sample line group, and a list of materials.

For information about creating a material list, see Generating Material Lists (page 1163).

# Mass Haul

Measure mass haul volumes and mark grade points, balance points, and free haul and overhaul volumes.

A mass haul (page 2477) diagram presents a graphical view of material moved in the proposed design site, displaying balance point (page 2468)s, free haul (page 2474) and overhaul (page 2479) regions, and grade point (page 2475)s. Mass haul diagrams are graphed with stations (sample lines) along the x-axis and cumulative material (typically earthworks) volume on the y-axis. The middle axis (zero cumulative volume) line is called the balance line (page 2468). When the mass haul line (page 2478) rises, it indicates that a project is in a cut region and when it descends, it is in a fill region. A mass haul region is marked with balance points and grade points.
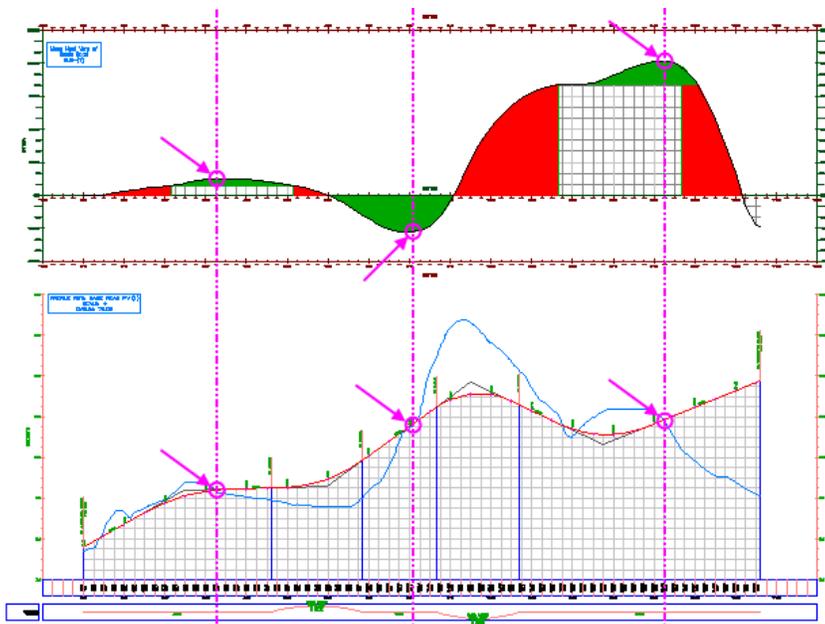


**Grade points (indicated by arrows and circles) in a mass haul diagram (above) and in a profile view connected by lines**

## Free Haul and Overhaul

Represent free haul and overhaul volumes using grade point and balance point methods.

During project development, contractors and design engineers may try to balance the mass haul requirements by considering free haul (page 2474) distances, any overhaul (page 2479), and the use of borrow pit (page 2469)s for embankments and dump site (page 2473)s for extra excavated volumes, based on the economic haul limits and the site conditions.

In AutoCAD Civil, you can specify the free haul distance as required. You can also specify the location of borrow pits and dump sites along the roadway, with assumed capacity of the borrow pits or dump sites.

There are two ways of representing free haul (and any overhaul) in mass haul diagrams in AutoCAD Civil:

■  measured from grade point (page 2475)s

■  measured from balance point (page 2468)s

## Measure from Grade Points

Represent free haul and overhaul from grade points along the mass haul line.
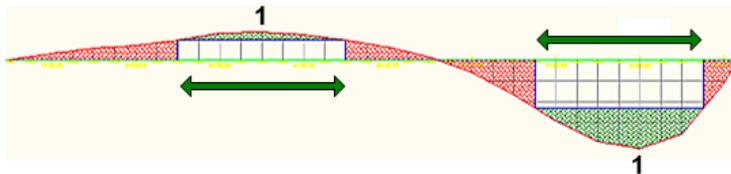


**Grade Point Method: The arrows indicate free haul distance, The number (1) indicates grade points. Freehaul volume is indicated in green (and by the grid between the green areas and the balance line) and overhaul volume is indicated in red.**

From grade point (page 2475)s, a segment the length of the free haul distance, which is parallel to the balance line (page 2468), is dropped so that the ends of the segment just touch the mass haul line (page 2478). The area enclosed within the segment and the mass haul line represents the free haul volume. Further, perpendicular lines from the ends of the segment are dropped so that they touch the balance line. The areas enclosed within these lines, the balance line, and the mass haul line represent the overhaul volume.

## Measure from Balance Points

Represent free haul and overhaul from balance points, where the mass haul line and balance line meet.
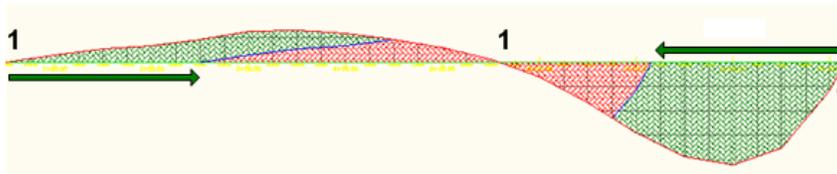


**Balance Point Method: The arrows indicate free haul distance. The number (1) indicates balance points. Freehaul volume is indicated in green and overhaul volume is indicated in red.**

In a mass haul diagram, the balance point (page 2468)s are located on the balance line, where the net volume is zero. In the balance points method of measuring free haul, the mass haul line is duplicated and shifted horizontally to the right (where the project transitions from cut to fill) or to the left (where the project transitions from fill to cut) by the free haul distance. The area enclosed by the two mass haul lines on the left if above the balance line, or on the right if below the balance line, represents the free haul volume. The remaining intersecting area represents the overhaul.

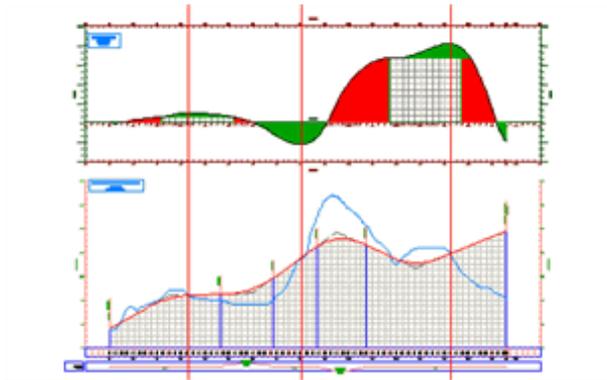# Creating and Editing Mass Haul Diagrams

Create a mass haul diagram using the Create Mass Haul Diagram wizard, and edit the mass haul diagram's properties, styles, and settings.

## Creating Mass Haul Diagrams

Create a mass haul diagram to display a partial or overall view of earthwork volumes.

Using the Create Mass Haul Diagram wizard (page 2301), you can create a mass haul diagram with a single click, accepting the default settings for name, style, display, and balancing options. Later, you can change the default settings in the Mass Haul Properties dialog box (page 2309). Alternatively, you can specify settings in the wizard.

You must have a material list for the drawing before you can create a mass haul diagram. A mass haul diagram (top) can be mapped directly to a profile view (bottom) created from the same alignment.



For information about how to create a material list, see Generating Material Lists (page 1163).

**To create a mass haul diagram**

1   Click Analyze tab ➤ Volumes And Materials panel ➤ Mass Haul.
    The Create Mass Haul Diagram wizard (page 2301) is displayed.

2   On the General page (page 2301), do any of the following:
    ■ Specify the alignment from which the mass haul diagram will be created.

    ■ Specify the sample line group from which the mass haul diagram will be created.

    ■ Specify a mass haul view style.

    ■ To edit the mass haul view style, next to Mass Haul View Style, click . (See Editing Mass Haul View Styles (page 1178).)

3   On the Mass Haul Display Option page (page 2301), do any of the following:
    ■ Select a material list.

    ■ Select a material to display as mass haul line (Total Volume, Total Cut Volume, Total Fill Volume, Total Unusable Volume, and any other material associated with the material list).

    ■ Specify a mass haul line style.

    ■ To edit the mass haul line style, next to Mass Haul Line Style, click . (See Editing Mass Haul Line Styles (page 1177).)

**4** On the Balancing Options page (page 2302), specify a free haul distance, and the type, location, and capacity of borrow pits and dump sites.

**5** Click Create Diagram.

**6** In the drawing, find a suitable location for the mass haul diagram. For ease of viewing, place the diagram outside the surface boundary, and above or below any section or profile views. Click that location to set the origin of the balance line at the left side of the mass haul diagram.

The mass haul diagram is displayed.

## Quick Reference

Ribbon

Analyze tab ➤ Volumes And Materials panel ➤ Mass Haul

Menu

Sections menu ➤ Create Mass Haul Diagram

Command Line

CreateMassHaulDiagram

Wizard

Create Mass Haul Diagram (page 2301)

## Editing Mass Haul Properties

Edit mass haul properties to specify naming, description and other parameters for the mass haul lines and views.

## Editing Mass Haul Line Properties

Balance mass haul volumes by editing mass haul line properties.

Edit mass haul line properties to specify a new name or description for the mass haul line, to change the free haul distance, and to add or remove borrow pits or dump sites.

**To edit mass haul line properties**

**1** To select a mass haul line and open the Mass Haul Line Properties (page 2309) dialog box, do either of the following:

■ In Toolspace, on the Prospector tab, expand the Sites collection ➤ <site name> ➤ Alignments collection ➤ <alignment name> ➤ Sample Line Groups collection ➤ <sample line group name> ➤ Mass Haul Lines collection. Right-click the mass haul line whose properties you want to edit. Click Properties.

■ In the active viewport, select the mass haul line whose properties you want to edit. Right-click and click Mass Haul Line Properties.

**2** On the Information tab of the Mass Haul Line Properties dialog box, do any of the following:

■ Change the name of the current mass haul line.

■ Enter a description of the current mass haul line.

■ Specify the mass haul line style.

- To edit the current mass haul line style, under Object Style, click  . For more information, see Editing Mass Haul Line Styles (page 1177).

**3** On the Balancing Options tab, you can do any of the following:

- Specify a new free haul distance.

- Add or remove borrow pits and dump sites, specifying their type, location and capacity.

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab ➤ <drawing name> ➤ Sites ➤ <site name> ➤ Alignments ➤ <alignment name> ➤ Sample Line Groups ➤ <sample line group name> ➤ Mass Haul Lines ➤ right-click <mass haul line name>. Click Properties.

Dialog Box

Mass Haul Line Properties (page 2309)

## Editing Mass Haul View Properties

Edit mass haul view properties to specify a new name or description.

**To edit mass haul view properties**

**1** To select a mass haul view and open the Mass Haul View Properties dialog box (page 2312), do either of the following:

- In Toolspace, on the Prospector tab, expand the Sites collection ➤ <site name> ➤ Alignments collection ➤ <alignment name> ➤ Sample Line Groups collection ➤ <sample line group name> ➤ Mass Haul Views collection. Right-click the mass haul view whose properties you want to edit. Click Properties.

- In the active viewport, select the mass haul view whose properties you want to edit. Right-click and click Mass Haul View Properties.

**2** On the Information tab of the Mass Haul View Properties dialog box, you can do any of the following:

- Change the name of the current mass haul view.

- Enter a description of the current mass haul view.

- To edit the current mass haul view style, under Object Style, click  . (For more information, see Editing Mass Haul View Styles (page 1178).)

**3** Click OK.

### Quick Reference

## Editing Mass Haul Styles

Edit mass haul style to improve the display of the mass haul line and the mass haul view.

## Editing Mass Haul Line Styles

Edit the mass haul line styles to improve the display of the mass haul and balance lines, and free haul and overhaul volumes.

**To edit mass haul line styles**

1  In the active viewport, select the mass haul line whose styles you want to edit. Right-click and click Edit Mass Haul Line Style.

2  On the Free Haul tab (page 2311) of the Mass Haul Line Style dialog box, specify how you want to show free haul in the mass haul diagram: measured from grade points (page 1173) or measured from balance point (page 1173).

3  On the Display tab (page 2311), do any of the following:
   ■  In Component Display, set visibility and color for the Mass Haul Line, Free Haul Line, Free Haul Area Hatch, and Overhaul Area Hatch.

   ■  In Component Hatch Display, set the pattern, angle and scale of the hatching for Free Haul Area Hatch, and Overhaul Area Hatch.

4  On the Information and Summary tabs, set administrative information.

5  Click OK to save the edits to the mass haul line styles.

### Quick Reference

### Editing Mass Haul View Styles

Edit the mass haul view styles to improve the display of the mass haul graph and its annotation.

**To edit mass haul view styles**

1 In the active viewport, select the mass haul view whose styles you want to edit. Right-click and click Edit Mass Haul View Style.

2 On the Information tab of the Mass Haul View Style dialog box, specify administrative information, such as the name of the style, and a description of the style.

3 On the Graph tab, specify the vertical and horizontal scale, and the view direction.

4 On the Grid tab (page 2314), specify how you want to show the grid.

5 On the Title Annotation tab, specify attributes of the text used to annotate the graph.

6 On the Horizontal Axes and Vertical Axes tabs, specify attributes of the major and minor ticks in the graph.

7 On the Display tab (page 2321), in Component Display, set the visibility and color for the graph.

8 In the Summary tab, modify information specified on other tabs.

9 Click OK to save the edits to the mass haul view styles.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab ➤ <drawing name> ➤ Sites ➤ <site name> ➤ Alignments ➤ <alignment name> ➤ Sample Line Groups ➤ <sample line group name> ➤ Mass Haul Views ➤ right-click <mass haul view name>. Click Properties. In the Mass Haul View Properties dialog box, Information tab, under Object Style, click  .

Dialog Box

Mass Haul View Style (page 2313)

# Editing Mass Haul Settings

Use mass haul settings to specify the default style and name format settings for mass haul lines and views and the default behavior for mass haul-related commands.

Settings are handled in a standard way throughout AutoCAD Civil. Access settings using the Toolspace Settings tree. Control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

On the Toolspace Settings trees, use the Mass Haul Line and Mass Haul View collection shortcut menus to establish defaults for all of the mass haul-related features and commands. You can change mass haul line- and mass haul view-specific settings at this level, or you can override the drawing ambient settings.

## Editing Mass Haul Line Settings

Edit mass haul line style settings to specify defaults for new mass haul diagrams.

**To create a new mass haul line style**

**1** In Toolspace, in the Settings tree, under Mass Haul Line, expand the Mass Haul Line Style collection.

**2** For the Mass Haul Line Style collection, do one of the following:
- Right-click the collection. Click New.

- Right-click the existing style to use as a template. Click Copy.

**3** In the Mass Haul Line Style dialog box (page 2310), enter or change the name of the style and other settings as required.

**4** Click OK.

**To edit a mass haul line style**

**1** In Toolspace, in the Settings tree, under Mass Haul Line, expand the Mass Haul Line Style collection.

**2** Right-click the style you want to edit. Click Edit.

**3** In the Mass Haul Line Style dialog box (page 2310), change the Free Haul or Display settings as required.

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Create: Settings tab: right-click Mass Haul Line collection ➤ New
Copy: Settings tab ➤ Mass Haul Line collection ➤ Mass Haul Line Styles collection: right-click <style name> ➤ Copy
Edit: Settings tab ➤ Mass Haul Line collection ➤ Mass Haul Line Styles collection: right-click <style name> ➤ Edit

Dialog Box

Edit Feature Settings - Mass Haul Line (page 2304)

## Editing Mass Haul View Settings

Edit mass haul view settings to specify default graph style and command settings for new mass haul diagrams.

**To create a new mass haul view style**

**1** In Toolspace, in the Settings tree, under Mass Haul View, expand the Mass Haul View Style collection.

**2** For the Mass Haul View Style collection, do one of the following:
- Right-click the collection. Click New.

- Right-click the existing style to use as a template. Click Copy.

**3** In the Mass Haul View Style dialog box (page 2313), enter or change the name of the style and other settings as required.

**4** Click OK.

**To edit a mass haul view style**

**1** In Toolspace, in the Settings tree, under Mass Haul View, expand the Mass Haul View Style collection.

**2** Right-click the style you want to edit. Click Edit.

**3** In the Mass Haul View Style dialog box (page 2313), change the Graph, Grid, Title Annotation, Horizontal Axes, Vertical Axes, or Display settings as required.

**4** Click OK.

**To edit mass haul view command settings**

**1** In Toolspace, in the Settings tree, under Mass Haul View, expand the Commands collection.

**2** Right-click CreateMassHaulDiagram. Click Edit Command Settings.

**3** In the Edit Command Settings - CreateMassHaulDiagram dialog box (page 2304), expand the Mass Haul Creation tree.

**4** Specify the Analyze Free Haul value as True.

**5** Specify the Free Haul Distance value or leave the default setting.

**6** Specify the Borrow Pit Capacity value or leave the default setting.

**7** Specify the Dump Site Capacity value or leave the default setting.

**8** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Create style: Settings tab: right-click Mass Haul View collection ➤ New
Copy style: Settings tab ➤ Mass Haul View collection ➤ Mass Haul View Styles collection: right-click <style name> ➤ Copy
Edit style: Settings tab ➤ Mass Haul View collection ➤ Mass Haul View Styles collection: right-click <style name> ➤ Edit
Edit command settings: Settings tab ➤ Mass Haul View collection ➤ Commands collection: right-click CreateMassHaulDiagram ➤ Edit Command Settings

Dialog Boxes

Mass Haul View Style (page 2313)
Edit Settings - Mass Haul View (page 2304)

# Material Analysis Command Reference

The AutoCAD Civil commands for material analysis-related functionality and a brief description of their functionality.

| Command | Description |
| --- | --- |
| AddMaterialVolumeTable | Creates a Material Volume Table (page 1168) |
| AddTotalVolumeTable | Creates a Total Volume Table (page 1168) |
| ComputeMaterials | Creates a new material list for a sample line group (page 1163) |
| CreateMassHaulDiagram | Creates a mass haul diagram (page 1174) |

| Command | Description |
| --- | --- |
| GenerateQuantitiesReport | Generates an external quantity takeoff report (page 1168) |

# Pipe Networks

# Pipe Networks

# 27

Use AutoCAD Civil pipe network features to draw 2D and 3D models of utility systems, such as storm sewers, sanitary sewers, and more.

You can create, edit, and delete pipe networks in plan view. You can also display pipe network parts in profiles and section views. Changes made to pipe networks in plan view are dynamically updated in any profile and section views.

Style-based rendering and labeling capabilities let you quickly create pipe networks with meaningful visual effects and data. For example, you can use many different styles to represent two-dimensional pipe networks in construction documents. You can also use visual rendering features to add realistic looking effects to three-dimensional pipe networks. Labeling features let you automatically add labels to identify specific types of pipes and structures within a pipe network.

AutoCAD Civil also has interference checking features let you quickly identify areas where pipes or structures physically collide, or are too close to one another.

You can also import or export pipe network data using Autodesk LandXML format, or analyze pipe network data through an API interface or with support from external, third-party analysis programs.

Pipe Networks Workflow

Pipe Networks Best Practices

## Understanding Pipe Networks

In AutoCAD Civil, a pipe network object serves as a container for managing pipe objects and structure objects that form a pipe network.
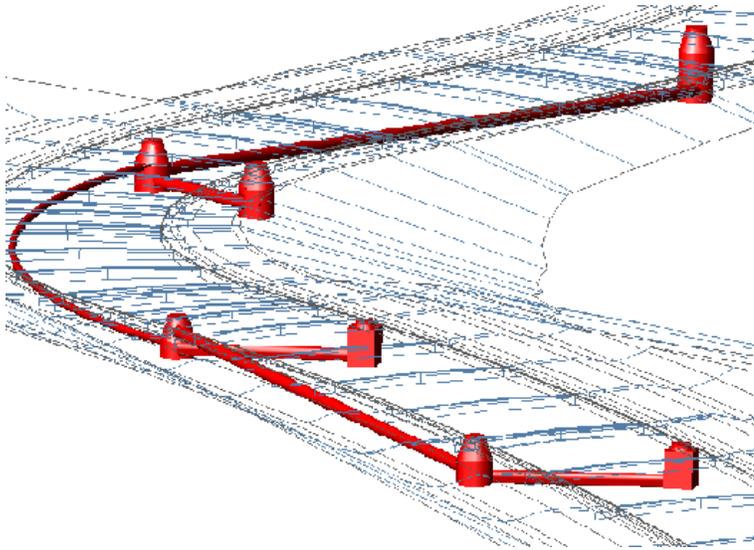
A pipe network can contain pipes only, structures only, or, more commonly, both pipes and structures. A part catalog provides access to pipe network items, such as circular, elliptical, or rectangular pipes, manholes, catch basins, and headwalls, and more.

After you have added a pipe network to your drawing, you can edit it in the following ways:

- Add, delete, change, swap, or move pipes or structures

- Resize pipes and structures

- Move a pipe network

- Create a parts list comprised of only the part catalog items needed for a particular project or network

- Edit elevations and sizes numerically or graphically

- Add, change, or delete labels on pipes, a span of connected pipes, structures, or on the entire pipe network

- Change the style of any pipe or structure in the pipe network

- Use styles to display pipes and structures according to their true shapes, or choose a symbolic representation, such as a block, for a structure

After the pipe network is created, you can extract a variety of data using LandXML export. For information, see LandXML Import and Export (page 1673).

Like other AutoCAD Civil objects, display styles for pipe network objects are based on styles assigned to the objects: specifically, the pipe objects and structure objects. Before creating pipe networks, you should be familiar with both creating and managing styles (page 53) and command settings (page 97).

## The Pipe Network Object

A pipe network object manages a collection of pipe objects and structure objects that are associated with each other to represent a pipe system.

Typically, the pipes and structures are connected to each other, forming a single pipe run or pipe network. The pipe and structure objects in a pipe network can be associated with a referenced alignment and or a surface, which provide them with station offset and elevation data. Each part in a pipe network can reference any given surface or alignment in the drawing.

The following sections describe the components that can comprise a pipe network:

**Pipe Network**

The pipe network object is used as the container object to associate pipes and structures that are part of the same pipe run or pipe network. A pipe network typically contains pipe objects and structure objects. Pipe network object names are displayed in the Prospector tree and in the Prospector list view.

**Pipe**

A pipe object is a drawing shape that represents straight or curved pipes used in utility networks, such as sewer and irrigation systems. In a drawing, the three-dimensional pipe shape is defined by: 1) the two-dimensional Part Shape (circular, elliptical, egg-shaped, or rectangular) of the pipe part that is selected from the part catalog; and 2) by specifying a linear path (for straight piped) or a curved path (for curved piped). Object names for pipes are not displayed in the Prospector tree. They are, however, displayed in the Prospector list view when you click Pipes under a pipe network in the Prospector tree.

**Structure**

A structure object is a drawing shape that is used to represent items, such as manholes, catch basins, and headwalls, used in utility networks. Structure shapes are inherently more complex than pipe shapes. In a drawing, the three-dimensional structure shape is defined by the definition of the structure part that is selected from the part catalog. Like pipes, object names for structures are not displayed in the Prospector tree. They are, however, displayed in the Prospector list view when you click Structures under a pipe network in the Prospector tree.

**Null Structure**

A null structure object is a special type of structure object that is inserted automatically when you connect a pipe directly to another pipe, with no structure shape between the two pipes. Just like other types of structure objects, object names for null structures are not displayed in the Prospector tree, but are displayed in the Prospector list view. You can assign a style to null structures so that they are not visible in the drawing.

**Part Catalog and Parts List**

AutoCAD Civil comes with a pipe network part catalog that contains a variety of pipe and structure shapes organized into part families and part sizes. Since the part catalog contains many items, you can create a parts list containing only the parts (pipes and structures) that you will use for a particular pipe network. Using a parts list saves you from having to navigate through the entire part catalog to find a desired part. For more information, see Part Catalog and Parts Lists (page 1237).

For overview information about AutoCAD Civil objects, see Understanding Objects (page 51).

# Pipe Network Styles and Display

Specify a variety of style and display characteristics for displaying pipes and structures in plan view, profile views, and section views.

When you first create a pipe network, the pipe and structure objects get their default style, render material, and rules from definitions in their referenced parts list.

After the pipe and structures are inserted into the drawing, the styles and rule settings for the pipe or structure object can be changed using either the Pipe Properties or the Structure Properties dialog box. You can also use the Pipe Network Vistas (page 2064) view to edit style or property values.

If you choose to run interference checks on pipe networks, you can set styles for displaying interferences.

The following table describes the ways you can specify styles for pipes and structures in a pipe network:

| To specify this style... | use the... |
| --- | --- |
| Default object style, rules, and render material for pipe and structure objects | Parts List definition. Specifically, you use the Pipes tab (page 2035) or the Structures tab (page 2036) of the Network Parts List dialog box. |
| Label style for pipe network object during creation | Create Pipe Network By Layout Dialog Box (page 2023) |
| Object style, rules, or render material for pipe and structure objects already in a drawing | Pipe Properties Dialog Box (page 2039) or Structure Properties Dialog Box (page 2051) |
| Display style for pipes or structures displayed in plan, profile, and section views already in a drawing | Pipe Style Dialog Box (page 2045) or Structure Style Dialog Box (page 2058). You can also override styles in Profile View Properties dialog box. |
| Display style for pipe network interferences | Interference Style Dialog Box (page 2074) |

In addition to specifying and editing object styles, display styles, rules, and render material, you can also add labels to pipe and structure objects either during creation or after the objects have been added to a drawing. For more information about pipe and structure labels, see Labeling Pipe Networks (page 1261).

For more information about styles, see Object Styles (page 53).

## Pipe Networks Collection (Prospector Tab)

Use the Pipe Networks collection in the Prospector tree to access the pipe networks and interference checks in a drawing or a project.

As pipe network objects are created, they are displayed in the Networks collection under the Pipe Networks collection. Expand the Networks collection to view the names of the pipe networks. Click Pipes or Structures to display a tabular list of the pipe network object informations in the Prospector list view. For more information, see The Toolspace Item View (page 82).

You can also view interference check results by expanding the Interference Checks collection. For more information, see Checking for Interferences (page 1226).

## Pipe Network Collection (Settings Tab)

Use the Pipe Networks collection in the Settings tree to manage pipe network settings, parts lists, interference styles, and command settings.

The Pipe Network object does not have any display styles. However, elements within the Pipe Network object, such as pipe objects and structure objects, have their own display styles. When you insert a pipe or structure object into a drawing, its default style comes from the part's style definition in the parts list. After the part is inserted into the drawing, you can change its style settings.

Right-click the Pipe Networks collection to do the following:

- Edit the pipe network feature settings. (page 1201)

- Refresh the display of the settings tree.

Expand the Pipe Networks collection to display and edit the command settings that are available for pipe networks.

## Pipe Collection (Settings Tab)

Use the Pipe collection in the Settings tree to manage pipe styles, pipe rule sets, pipe label styles, and pipe table styles.

| For information about ... | See... |
| --- | --- |
| pipe styles | Pipe Network Styles and Display (page 1187) |
| pipe rule sets | Part Rules (page 1247) |
| pipe label styles | Labeling Pipe Networks (page 1261) |
| pipe table styles | Adding Pipe Tables (page 1271) |

## Structure Collection (Settings Tab)

Use the Structure collection in the Settings tree to manage structure styles, structure rule set styles, structure label styles, and structure table styles.

| For information about ... | See... |
| --- | --- |
| structure styles | Pipe Network Styles and Display (page 1187) |
| structure rule sets | Part Rules (page 1247) |
| structure label styles | Labeling Pipe Networks (page 1261) |
| structure table styles | Adding Structure Tables (page 1272) |

## Common Part Terms

This section provides descriptions for some terms that are commonly used when referring to pipe network parts.

**Structure Terms**

■ Rim. The top elevation of a manhole or catch basin (junction) structure.

■ Frame. The support for the cover on a manhole or grate for a catch basin. The top elevation of the frame, cover, and grate typically coincides with the insert (rim) elevation.

■ Cover. The top access cover of a manhole, used for servicing or inspecting the manhole. Covers are typically circular in shape, but they can also be triangular, so that they do not fall into the manhole when removed. The cover sits in the frame and is usually constructed of cast iron.

■ Grate. The top opening of a catch basin that typically contains numerous openings to allow surface water entry into the structure. The grate sits in the frame and is usually constructed of cast iron. Grates can be various shapes but are typically circular or square.

■ Sump and sump depth. Sump refers to the bottom area or chamber on the inside of a junction structure, such as a catch basin. The sump area is sometimes used as a reservoir to catch debris. Sump depth refers to the distance from the bottom of the inside of the structure to the lowest invert of all pipes connected

to it. The sump depth for a particular type of structure can vary according to design requirements. Manholes typically have no sump depth.

The following illustration shows the locations of components on a typical junction structure.
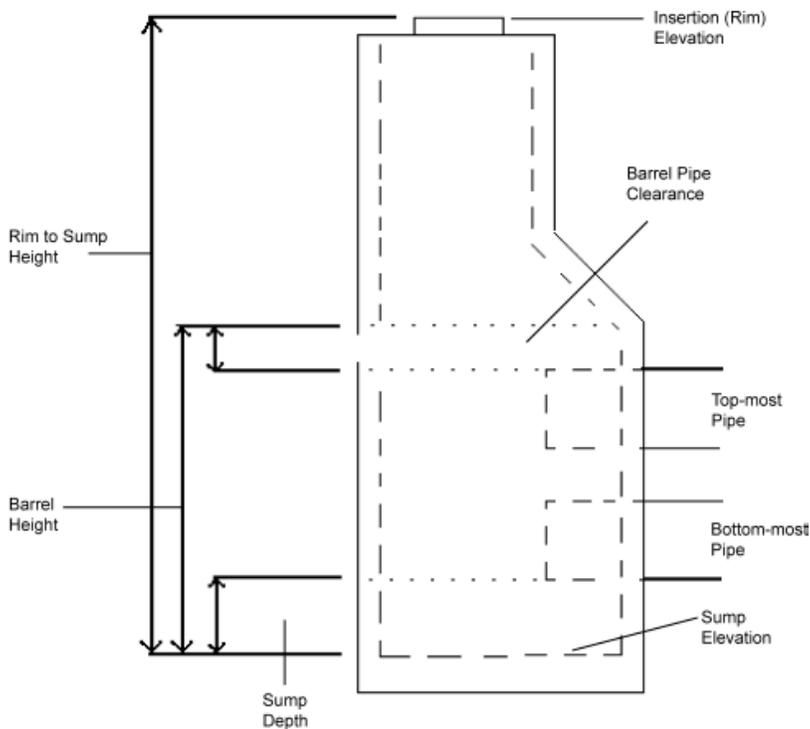


**Pipe Terms**

■ Invert. The elevation of the inside bottom of a pipe. Also referred to as flowline.

■ Crown. The elevation of the inside top of a pipe.

■ Pipe centerline elevation. The elevation of the centerline of a pipe.

■ Cover or Depth. Refers to the depth of the material above the topmost outer part of a pipe.

■ Slope. When slope is used to refer to the slope of a pipe, it is a measure of steepness. For pipes, the slope is defined by rise (or fall) over run, where rise is the difference in elevation along a run of pipe length, measured horizontally. Slope is typically represented in percent. For example, a slope of pipe at 2% means that for every 100 feet of horizontal length of pipe, the pipe rises 2 feet. In AutoCAD Civil, the slope can be represented in a variety of ways in dialog boxes and in labels in the drawing.

## Hydraulics and Hydrology Feature Overview

You can display hydraulics and hydrology properties in AutoCAD Civil pipe networks.

External applications can be used to add hydraulics and hydrology property information to AutoCAD Civil pipe networks. It is also possible to add hydraulics and hydrology property information directly to AutoCAD Civil pipe networks using the Pipe Network Vista (Panorama window). For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

For more information about how to use these extensions, click Help on the menu bar of the application. The Help menu provides access to the online Help and PDF user's guide provided with each extension.

For information about using AutoCAD Civil to perform hydraulics and hydrology tasks, see Using Hydraulics and Hydrology Features (page 1274).

# Creating Pipe Networks

There are several ways you can create pipe networks.

For example, you can create a pipe network using the Network Layout Tools toolbar, or you can select an existing polyline or feature line in a drawing and create a pipe network from it. You can also use import features to import existing pipe networks into your drawing.

**Importing Pipe Data**

The following table describes the import features available for pipe networks.

| Command | Description |
| --- | --- |
| Import Data From Land Desktop | You can use the Import Data From Land Desktop command (ImportLDTData) to import pipe run data from AutoCAD Land Desktop into AutoCAD Civil. When you import pipe data, each continuous pipe run in AutoCAD Land Desktop is imported as a new, single pipe network object in AutoCAD Civil. For more information, see Importing LandXML Drawing Data (page 1681). |
| LandXML Import | You can export pipe run data from AutoCAD Land Desktop using the LandXML Export command (LandXMLOut) and then import it into AutoCAD Civil using the LandXML Import command (LandXMLIn). For more information about these commands, see LandXML Import and Export (page 1673). |
| ImportStormSewerData | Using this command, you can create AutoCAD Civil pipe networks by importing pipe networks that were created in the Storm Sewers Extension from an *.stm* file. For more information, see Importing Pipe Network Data from an .stm File (page 1278). |

Before importing Land Desktop pipe run data, make sure the Parts List you associate with the pipe network has part sizes that make sense for the pipe run you will be importing. See Part Catalog and Parts Lists (page 1237) for more information.

**Managing, Displaying, and Customizing Pipe Networks**

All pipe networks, once created, are managed in the same way, regardless of the method used to create them. All pipe networks are listed in the Pipe Networks ➤ Networks collection in the Prospector tree.

Before creating a pipe network, it can be useful to first create the underlying data used by the pipe network, such as surfaces, alignments, profiles, section views, and your own custom parts lists. However, you do not have to have these components in place to create a pipe network.

| For information about creating... | See... |
| --- | --- |
| Surfaces | Creating Surfaces (page 601) |

| For information about creating... | See... |
|---|---|
| Alignments | Creating Alignments (page 887) |
| Profiles | Creating Layout Profiles (page 1048) |
| Section Views | Creating/Editing Section Views (page 1153) |
| Parts Lists | Part Catalog and Parts Lists (page 1237) |

When you create a pipe network, the default settings come from the selections you make in the Create Pipe Network dialog box, and from the part definitions specified in the part catalog. After the pipe and structure objects are in your drawing, you can change many of the settings for any new parts created in the network. For information about how to change pipe network default settings, see Changing Pipe Network Default Style Settings (page 1202).

## Creating a Pipe Network Using the Layout Tools

You can create a pipe network using the Network Layout Tools.

To create a new pipe network using the layout command, you must first enter a unique name for the pipe network on the create dialog box.

To automatically reference station and elevation data for the pipe network, you can associate the pipe network with an alignment and a surface.

You can also set label styles and specify automatic labeling as new parts are created. After making these choices or accepting the defaults, you can use the Network Layout Tools to draw the pipe network.

**To create a pipe network using the layout tools**

1   Click Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools 🔧 , or in the Prospector tree, expand the Pipe Networks collection, then right-click the Networks collection, and click Create Pipe Network By Layout.

2   In the Create Network (page 2023) dialog box, in the Network Name field, enter a name for the pipe network.

   To name the pipe network, select a default name from the naming template, or enter a new name. For more information, see Name Template Dialog Box (page 1806).

3   In the Network Description field, enter an optional text description for this pipe network.

4   In the Network Parts List field, select a parts list or accept the default parts list for this pipe network. For more information, see Part Catalog and Parts Lists (page 1237).

5   To view or change the default layers that will be assigned to various pipe network parts as they are created, click Layers to open the Network Layers (page 2063) dialog box.

6   To reference a surface and/or an alignment, select the item from the list, or click to select a surface and/or alignment in the drawing.

7   Optionally, you may select label styles for pipes and structures added to the pipe network. When <none> is specified, no labels are added to the items, but you can add labels later if desired. See Labeling Pipe Networks (page 1261) for more information.

8   Click OK in the Create Pipe Network dialog box. The Network Layout Tools toolbar is displayed, and the pipe network name is displayed in the Pipe Networks collection on the Prospector tab.

**9** On the Network Layout Tools toolbar, select the desired parts (pipes and structures) in the Pipe List and Structure List. For example, select 48-inch Concentric Catch Basin for a structure, and 12-inch Concrete Pipe 1 US Imperial for a pipe. For more information, see Part Catalog and Parts Lists (page 1237).

**10** Select the types of pipe network objects you want to insert by clicking the Down Arrow button next to Draw Pipes And Structures and selecting one of the following:

- Pipes and Structures. Inserts pipes and structures during the same command operation. This is the default command when creating a new pipe network.

- Pipes Only. Inserts just one or more pipes (without inserting structures) into the pipe network.

- Structures Only. Inserts just one or more structures into the pipe network, without inserting any pipes.

Pipes and Structures is the default active layout command. The initial prompt (by default) upon entering the layout command is `Specify the structure insertion point:`

Upon reopening the layout toolbar, the default command is the last command that was used.

**11** Click the Toggle Upslope/Downslope button to set the direction of the pipe networks part(s) to either a positive slope value or direction (upslope) or a negative slope value or direction (downslope). This button acts as a toggle. Downslope is the default. When the downslope icon is displayed, each new pipe that is inserted will slope downstream from the previous part. When the upslope icon is displayed, each new pipe inserted will slope upstream from the previous part.

**12** Begin drawing the pipe network by specifying the insertion point for the first part (pipe or structure) of the pipe network.

**NOTE** If you are in Draw Pipes And Structures mode, the first click defines the first structure insertion point, as well as the starting point of the first pipe object. The second click specifies the end of the first pipe object, and inserts a new structure object connected to that pipe. Subsequent clicks specify insertion points for subsequent pipes and structures until you end the command by pressing Enter. If you are in Draw Pipes Only mode, each click specifies the insertion point for a pipe end (a pipe start point or end point). If you are in Draw Structures Only mode, each click specifies the insertion point for a structure.

**13** Specify the next insertion point in the drawing.

**14** When you finish drawing the pipe network, press Enter to end the command.

After you create a pipe network, you may want to refine parts of it. For example, you may want to add a new pipe or structure, remove parts, move parts, or add labels. For information, see Editing Pipe Networks (page 1203).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools

Menu

Pipes menu ➤ Create Pipe Network By Layout

Command Line

CreateNetwork

Dialog Box

## Creating Pipe Networks from Objects

Create pipe networks from existing objects in your drawing.

Using this method, you can automatically create a pipe network from entities such as lines, arcs, 2D and 3D polylines, feature lines, or alignments.

The following table lists the types of entities you can use to create pipe networks:

| Entity | Description |
| --- | --- |
| Line | Creates a pipe network from a line that you selected. The pipe network is inserted as a straight pipe with structures at each end. |
| 2D polyline | Creates a pipe network from lightweight or heavyweight 2D polylines. |
| 2D spline polyline | Creates a series of straight line segments using fit points to define pipe endpoints and structure locations. For more information about fit points, see the AutoCAD Help. |
| 3D polyline | Creates a pipe network using vertices in 3D to define pipe endpoints and structure locations. |
| Arc | Creates a pipe network from an arc that you selected. The pipe network is inserted as a curved pipe with structures at each end. |
| Feature line | Creates a pipe network from a feature line. |
| Alignment | Creates a pipe network from an alignment. |

You can select objects in the current drawing or from an Xref. You can create a pipe network that includes the types of pipes and structures you specify.

This method of creating a pipe network uses the Pipes and Structures insertion method. You cannot use the Pipes Only or Structures Only insertion mode with this command. If you want to create a single pipe, or a single structure using this command, create the pipe network with pipes and structures, and then delete the desired parts.

For 3D entity types, you can select the Use Vertex Elevations option on the Create Pipe Network From Object dialog box. When this option is selected, the elevations of any vertices along the selected entity are used to set the elevations of the pipes created in the network. For 3D entity types, this option determines if the 3D vertex elevations are honored by the pipes created. When checking this option, rules are not applied as they would likely be in conflict with the vertex elevations

**To create a pipe network from an object**

1  Click Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Create Pipe Network From Object ![icon] .

2  In the drawing, select the object.

**NOTE** You can enter Xref to select an object from an Xref.

You may be prompted to reverse flow direction. Press Enter or click OK to keep the current flow direction. Click Reverse to change the current flow direction. Flow direction arrows indicate the direction of the flow.

**3** In the Create Pipe Network From Object (page 2024) dialog box, specify a name for the pipe network.

**4** In the Description field, enter an optional description for the pipe network.

**5** In the Network Parts List field, select a parts list or accept the default parts list for this pipe network. For more information, see Part Catalog and Parts Lists (page 1237).

**6** In both the Pipe To Create and Structure To Create fields, select the types of pipes and structures to be used.

**NOTE** You must select both a pipe and a structure type. After creating the pipe network, if desired, you can delete parts you want removed.

**7** To view or change the default layers that will be assigned to various pipe network parts, click Layers to open the Pipe Network Layers (page 2063) dialog box.

**8** To reference a surface and/or an alignment, select the item in the list, or click to select a surface and/or alignment in the drawing.

**9** If you want the originally selected object to be removed from the drawing automatically, clear the Erase Existing Entity check box.

**10** Select the Use Vertex Elevations check box if you want to use the elevations of any vertices along the selected entity to set the elevations of the pipes created in the network.

**NOTE** For 3D entity types, this option determines if the 3D vertex elevations are honored by the pipes created. When checking this option, rules are not applied as they would likely be in conflict with the vertex elevations.

**11** Click OK in the Create Pipe Network From Object dialog box.

The new pipe network is displayed in the drawing, and the pipe network name is displayed in the Pipe Networks collection on the Prospector tab.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Create Pipe Network From Object

Menu

Pipes menu ➤ Create Pipe Network From Object

Command Line

CreateNetworkFromObject

Dialog Box

Create Network From Object

# Creating Alignments from Pipe Network Parts

Use the Create Alignment From Network Parts command to create an alignment from existing pipe network parts.

Using this method, you can automatically create alignment objects from existing pipe network parts (pipes and or structures) in your drawing.

You can also automatically create a profile and/or profile view showing the parts when you create the alignment.

When you use this command, the new alignment is displayed in the drawing and in the top-level Alignments collection or specified Site on the Prospector tab. The pipe network parts you selected to create the alignment are unaffected by the creation of the alignment.

**To create an alignment from pipe network parts**

1  Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Network Parts 

2  Select the first pipe network part (pipe or structure) to be used as the starting point for the alignment.

3  Select the next pipe network part to create a path for the alignment, or press Ctrl+Z to undo the previous path specified.

4  In the Create Alignment From Network (page 2025) dialog box, specify a Site or accept the default <None>. See Alignment and Site Interaction (page 721) for more information.

5  Enter a unique name for the alignment.

6  Enter an optional description for the alignment.

7  Specify an alignment style or accept the default style.

8  Specify the object layer settings.

9  Specify an alignment label set or accept the default label set.

10  Optionally, click Create Profile And Profile View to create a profile and a profile view from this alignment.

11  Click OK.

12  If you selected Create Profile And Profile View, the Create Profile View dialog box is displayed. Create the profile view and click OK.

The alignment name is displayed in the top-level Alignments collection or specified Site on the Prospector tab. The pipe network parts you selected to create the alignment are unaffected by the creation of the alignment.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Network Parts 

Menu

Pipes menu ➤ Utilities ➤ Create Alignment From Network Parts

Command Line

CreateAlignFromNetwork

## Drawing Straight Pipes

Draw straight pipes in a pipe network using the Network Layout Tools.

You can draw straight pipes when you are using the Draw Pipes And Structures insertion mode or when you are using the Pipes Only insertion mode.

After they are inserted, you can edit straight pipes using grips.

---

**TIP** You can use transparent commands for pipe network objects when you are prompted to specify radius, length, or station offset. For more information, see Using Transparent Commands Within a Running Command (page 1582).

---

**To draw a straight pipe in a pipe network**

1 Click Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools 🗝️ to create a new pipe network, or Pipe Networks tab ➤ Modify panel ➤ Edit Pipe Network 📝 to select an existing pipe network.

2 On the Network Layout Tools toolbar, click or 🔩 .

3 Specify the start point for the pipe.

4 Specify the end point.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools 🗝️

Menu

Pipes menu ➤ Create Pipe Network By Layout or Pipes menu ➤ Edit Pipe Network

Command Line

EditNetwork

Dialog Box

Network Layout Tools (page 2031)

## Drawing Curved Pipes

Draw curved pipes in a pipe network using the Network Layout Tools.

You can draw curved pipes when you are using 🔩 Draw Pipes And Structures insertion mode or when you are using the Pipes Only insertion mode.

No matter which insertion mode you are using, the resulting curved geometry is equivalent to the AutoCAD arc object, where the insertion point (or start point) of the pipe, the midpoint, and the endpoint of the curved pipe can be gripped to edit the arc.

Curved pipe objects do not have persistent tangency once the curved pipe is created. Therefore, once a curved pipe is grip edited, any tangency to adjacent pipes may be lost.

There are three ways that you can create curved pipes. Each of these methods are similar to methods used for creating alignment curves, or AutoCAD arc objects.

**Curve Creation Modes**

You can use the default curve creation mode when you are using the Pipes and Structures insertion mode, or when you are using the Pipes Only insertion mode. If you already have a structure inserted, and you are adding a pipe to it, clicking on the structure specifies the initial direction and start point for the curved pipe. You are then prompted for an end point.

This method is used when the curved pipe is starting from a structure (or a null structure) with only one pipe already connected to it. With this method, the initial start point for the curved pipe comes from the structure. The start direction comes from the one pipe that is already connected to that structure. This ensures that the curved pipe will be tangent to the last pipe drawn, or to a pipe that already exists at the structure (or null structure) from where the curved pipe is starting.

If no pipe is found to get a direction from, you are prompted to specify a direction at the first point.

**Specifying Two Points and a Radius**

Using this curved pipe creation mode, you specify the two endpoints of the curve and a radius. This method also lets you choose the curve direction (clockwise or counter-clockwise).

**Specifying Three Points**

Using this curved pipe creation mode, the first point is selected, then a point along the curve, then the curve endpoint.

---

**TIP** You can use transparent commands for pipe network objects when you are prompted to specify radius, length, or station offset. For more information, see Using Transparent Commands Within a Running Command (page 1582).

---

**To draw a curved pipe using the default curve creation mode**

1 On the Network Layout Tools toolbar, click ⬭ Pipes Only or ⬭ Pipes and Structures.

2 If you clicked Pipes and Structures mode, specify the insertion point for the structure.
  If you clicked Pipes Only mode, specify the first pipe point.

3 Enter c (curve), and press Enter to begin specifying the curve.

4 Specify the second point of the curve.

5 Specify the end point for the curve.

6 Press Enter to end the command.

**To draw a curved pipe by specifying two points and a radius**

1 On the Network Layout Tools toolbar, click ⬭ Pipes Only or ⬭ Pipes and Structures.

2 If you clicke ⬭ Pipes and Structures mode, specify the insertion point for the structure.
  If you clicked Pipes Only mode, specify the first pipe point.

3 Enter c (curve), and press Enter.

4 Enter r (radius) and specify a radius.

**5** Specify a curve direction, either clockwise or counter clockwise.

**6** Specify the end point for the curved pipe.

**7** Press Enter to end the command.

**To draw a curved pipe by specifying three points**

**1** On the Network Layout Tools toolbar, click � Pipes Only or 🗃 Pipes and Structures.

**2** If you clicked 🗃 Pipes and Structures mode, specify the insertion point for the structure. If you clicked Pipes Only mode, specify the first pipe point.

**3** Enter c (curve), and press Enter.

**4** Specify the second point of the curve.

**5** Specify the end point for the curved pipe.

**6** Press Enter to end the command.

## Quick Reference

Ribbon

Click Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools 🗃

Menu

Pipes menu ➤ Create Pipe Network By Layout or Pipes menu ➤ Edit Pipe Network

Command Line

EditNetwork

Dialog Box

Network Layout Tools (page 2031)

# Creating a Pipe Network Reference

You can create a reference to an existing pipe network in a data shortcut (page 2472) or a Vault project.

The pipe network reference is a lightweight read-only copy of the original, but it gives you access to the network design data. Before you create the pipe network reference, it must exist in the current project collection on the Toolspace Prospector tab.

**See also:**

■ Creating Data Shortcuts (page 131)

■ Vault: Checking a Drawing in to a Project (page 158)

**To create a reference to a data shortcut pipe network**

**1** In Toolspace, on the Prospector tab, in Master View, ensure that the correct working folder is identified on the Data Shortcuts node.

**2** Expand the Data Shortcuts ➤ Pipe Networks collection, right-click the desired network, and then click Create Reference.

The Create Pipe Network Reference dialog box is displayed, in which you can optionally change properties of the reference network, as described in the following steps.

**3** In the Create Pipe Network Reference dialog box, to change the source of the reference, select the source in the Source Pipe Network drop-down list.

**4** Enter a unique name for the pipe network and optionally, a description.

**5** Specify a parts list and object layer.

**6** Specify whether to override reference surface or alignment or select the defaults.

**7** Select to include, or not include, source drawing labels or select other label styles.

**8** Click OK to create the pipe network.

The pipe network name is displayed under the Pipe Networks collection in the Prospector tree with a ▣ next to it.

**To create a reference to a pipe network in a Vault project**

**1** In Toolspace, on the Prospector tab, in Master View, expand the Projects ➤ <project name> ➤ Pipe Networks collection, right-click the desired network, and then click Create Reference.

The Create Pipe Network Reference dialog box is displayed, in which you can optionally change properties of the reference network, as described in the following steps.

**2** In the Create Pipe Network Reference dialog box, to change the source of the reference, select the source in the Source Pipe Network drop-down list.

**3** Enter a unique name for the pipe network and optionally, a description.

**4** Specify a parts list and object layer.

**5** Specify whether to override reference surface or alignment or select the defaults.

**6** Select to include, or not include, source drawing labels or select other label styles.

**7** Click OK to create the pipe network.

The pipe network name is displayed under the Pipe Networks collection in the Prospector tree with a ⤴ next to it.

## Quick Reference

Toolspace Right-Click Menu (for data shortcut project)

Prospector tab: Data Shortcuts ➤ Pipe Networks ➤ <network name> ➤ Create Reference

Toolspace Right-Click Menu (for Vault project)

Prospector tab: Projects ➤ <project name> ➤ Pipe Networks ➤ <network name> ➤ Create Reference

Command Line

CreateNetworkReference (for Vault pipe network only)

Dialog Box

Create Pipe Network Reference Dialog Box (page 2200)

## Resetting Pipe Network Reference Labels

Use the Reset Network Labels command to reset a referenced network's labels to match the source drawing's labels.

You can make changes to reference network labels after referencing a pipe network into the current drawing.

Use this command when you have created a referenced pipe, and you want the labels in the referenced pipe to match the source drawing's labels, rather than the current drawing's labels.

For information about creating a pipe network reference, see Creating a Pipe Network Reference (page 1199).

**To reset a pipe network reference's labels**

1 Select the pipe network. Click Pipe Networks tab ➤ Labels & Tables panel ➤ Reset Labels .

2 Select a referenced pipe network part. The pipe network's labels are deleted and replaced with labels from the source drawing. If you select parts from multiple referenced pipe networks, changes apply to all of the referenced pipe networks.

**NOTE** After selecting this command, if there are no referenced pipe networks in the current drawing, an error message is displayed. If a source drawing is not found, an error message is displayed. If you select a part that is not from a referenced pipe network, you are prompted to select a referenced pipe network part.

### Quick Reference

Ribbon

Pipe Networks tab ➤ Labels & Tables panel ➤ Reset Labels 

Menu

Select the pipe network. Pipes menu ➤ Utilities ➤ Reset Network Labels

Command Line

ResetNetworkLabels

## Editing Pipe Network Settings

Use pipe network settings to specify the default styles and behavior for pipe network commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Toolspace Settings tab. You can control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

In the Settings tree, you can use the Pipe Network collection, the Pipe collection, or the Structure collection to establish defaults. Use the Pipe Network collection to specify defaults for all pipe network commands. You can change object-specific settings at this level, or override the drawing ambient settings.

**NOTE** Overrides to the drawing ambient settings at the Pipe Network collection level, the Pipe collection level, the Structure collection level, and the Pipe Network Commands collection level affect only the specified level. The drawing level settings are not changed.

The topics in this section describe only those settings that affect pipe networks and pipe network commands. For more information about the drawing ambient settings for the Pipe Network, Pipe, and Structure collections, see Specifying Ambient Settings (page 68).

# Changing Pipe Network Creation Settings

Use the pipe network creation and naming settings in the Pipe Network Settings (page 2026) dialog box to change pipe network settings before you create a pipe network.

For example, before creating a pipe network, you may want to specify the default rules used for pipes or structures in the parts list when new parts are added to it.

Note that if a closed lock appears in the Lock column for a property, the property is locked at a higher level of the Settings tree. It cannot be changed at this level.

If you change a property value, a check mark is placed in the My Override column. This override is also noted in the property table for related objects higher up the Settings tree, where a green arrow appears in the Child Override column.

**To change pipe network creation settings**

1 Do one of the following:

■ **To edit settings for all pipe network commands**: In Toolspace, on the Settings tab, right-click the Pipe Network collection. Click Edit Feature Settings.

■ **To edit settings for a specific command**: In Toolspace, on the Settings tab, expand the Commands collection in the Pipe Network collection. Right-click the name of the command for which you want to change settings. Click Edit Command Settings. For example, right-click the CreateNetwork command and click Edit Command Settings.

The Edit Feature Settings (page 2026) dialog box is displayed.

2 To edit the default naming format for pipe networks and data that can be extracted from the pipe network, expand the Default Name Format property group. Edit the Name Template settings.

3 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Pipe Network collection ➤ Edit Feature Settings
OR
Settings tab: Pipe Network ➤ Commands ➤ Right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings (page 2026)

# Changing Pipe Network Default Style Settings

Use the pipe network settings to specify default object styles and labels for pipe network creation and for data that you create or extract from a pipe network.

The default styles specified in these settings are used to establish the default setting for the style in the Create Pipe Network and Pipe Network Properties dialog boxes.

**To change the pipe network style and label default settings**

1 In Toolspace, on the Settings tab, right-click the Pipe Network collection. Click Edit Feature Settings to open the Pipe Network Settings (page 2026) dialog box.

2 Expand the ▒ Default Styles property group.

**3** To specify a default style for pipe network objects, such as pipes and structures, or for render material, click the corresponding field. Click ⊡ to display the default style dialog box, where you can select a style.

**4** To specify a default label style for pipe network components in plan and profile views, click the corresponding field. Click ⊡ to display the label style dialog box, where you can select a label style.

**5** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Pipe Network collection ➤ Edit Feature Settings

Dialog Box

Pipe Network Settings (page 2026)

# Editing Pipe Networks

You can edit pipe networks using the Network Layout Tools toolbar, pipe network vistas, or grips.

## Editing Pipe Network Using the Network Layout Tools

You can use the network layout tools toolbar to edit a pipe network.

Editing commands are the same commands you use for creating a pipe network.

**To edit a pipe network using the layout tools**

**1** In the drawing, right-click a part in a pipe network. Click Edit Network.

**2** Click a button in the Network Layout Tools (page 2031) dialog box to add, change, or delete pipe network parts, or change the surface, alignment or parts list that are referenced.

### Quick Reference

Ribbon

Select pipe network. Click Pipe Networks tab ➤ Modify panel ➤ Edit Pipe Network 📝

Menu

Pipes menu ➤ Edit Pipe Network

Object Shorcut Menu

Edit Network

Command Line

EditNetwork

Dialog Box

Network Layout Tools (page 2031)

## Editing Pipe Networks Using the Pipe Network Vistas

Use the Pipe Network Vistas to display all the components that are part of the pipe network.

The Pipe Network Vistas display pipe network data in a form that is similar to a spread sheet. Each row represents a pipe network part. Each column represents an attribute for a pipe network part. There is a tab to view pipe objects and a tab to view structure objects. The purpose of this view is to let you quickly see all the parts that make up the pipe network, along with their associated properties. Right-click a column heading to display options for customizing the display of detail in the column views.

**See also:**

■ The Toolspace Item View (page 82)

■ The Panorama Window (page 102)

**To edit a pipe network using the Pipe Network Vistas**

1 Select a pipe network part in the drawing. Right-click the part and click Edit Network.

2 On the Network Layout Tools toolbar, click 🖳 .

> **NOTE** You can edit only values that are available, you cannot edit data in disabled (shaded) cells.

### Quick Reference

Object Shorcut Menu

Edit Network ➤ Pipe Network Vistas 🖳

Dialog Box

Pipe Network Vistas (page 2064)

## Editing Pipe Networks Using Grips

Use grips to move, shorten, or stretch pipe or structure objects, or to change the size of a pipe.

Some of the grip editing behavior for pipes and structures is similar to the grip editing behavior for alignments. For more information about basic grip editing behavior for alignments, see Editing Alignments Using Grips (page 996).

**Grip Editing Pipes**

When moving a pipe in a pipe network using grips, structures connected to the pipe are no longer connected to the pipe, unless the pipe end being gripped is moved to another location within the network.

You disconnect the pipe from a structure if the end is attached to a structure. If you grip the end of the pipe to another structure, you can connect to it when a glyph appears next to the grip.
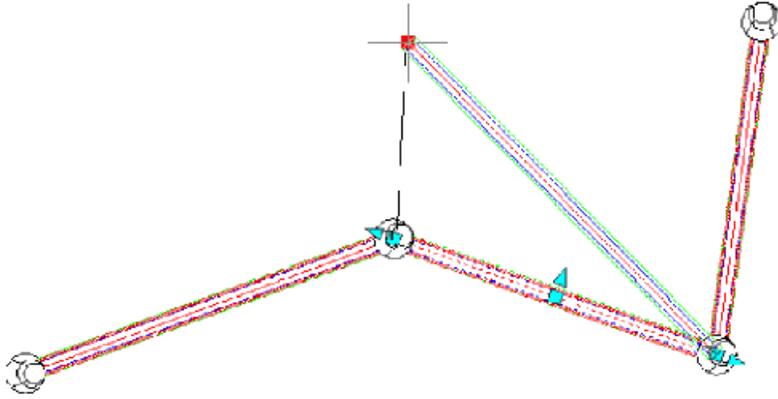
Grip editing a pipe that is connected to a structure does not move the structure along with it. Depending on the rim and sump behaviors of the structure, it may be necessary to adjust the structure vertically to accommodate a new sump elevation, or top of barrel, when the pipe is disconnected from the structure.

### Grip Editing Structures

When the location of a structure is edited, all the pipe ends connected to it move, along with the structure. The elevations of the original pipe ends connected to the structure are maintained.

### Pipes and Elevation Data

When grip editing a pipe, if no three-dimensional object snap is used (any value other than zero), the elevation of the end of the pipe is maintained when the pipe is connected to a new structure.

When a pipe is grip edited and moved over an existing pipe that it will "break," the end of the pipe that is being grip edited moves to the elevation of the pipe that is being broken.

The lengthening grips at the ends of the pipes extend the pipe end in a 2D plan view while maintaining the slope of the pipe. A 3D snap does not affect the elevations of the pipe when using the lengthening grip or the resize grips.

### Snapping to 3D

When viewing and editing pipe networks in plan view, you can quickly edit not only the two-dimensional geometry of the pipe, but you can also snap the pipe to three-dimensional locations to affect the elevation of either end of a pipe, or both ends simultaneously.

### Endpoint Free Grip

In plan view, clicking the endpoint free grip moves the selected endpoint of the pipe to a new specified point. The midpoint and opposite endpoint of the pipe are held during this edit. On a curved pipe, the behavior is identical to AutoCAD arc object behavior.
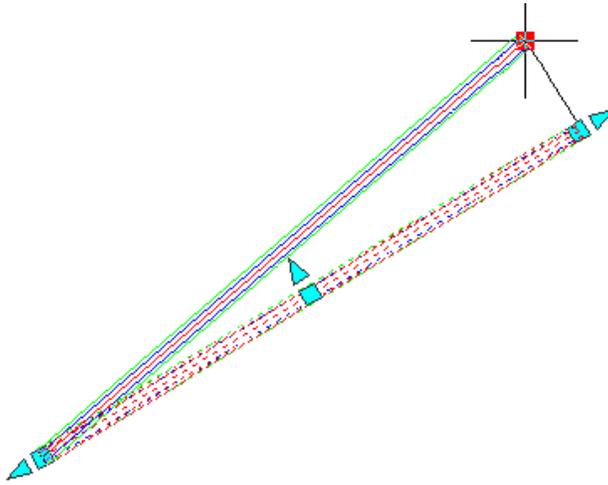
**Midpoint Free Grip**

This moves the entire pipe to a new specified point. On a curved pipe, the midpoint grips behaves just like AutoCAD arc objects: the endpoints are held, and the cursor acts as a pass-through location.
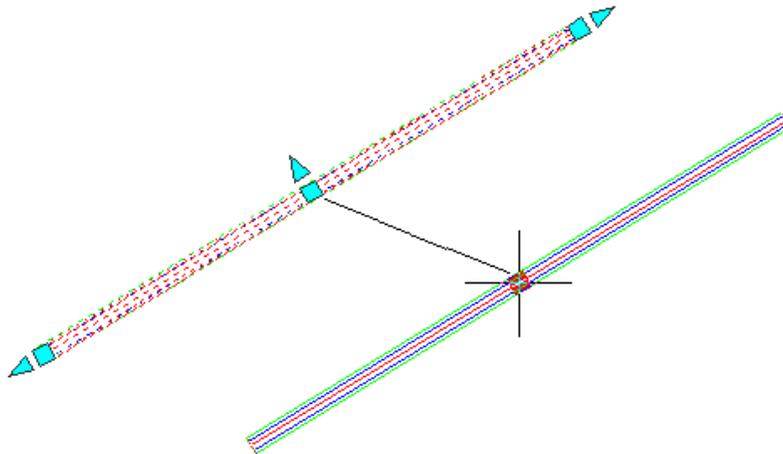


**Length Constraint Grip**

A pipe can be made shorter or longer by gripping the end of the pipe. Only the end that is gripped shortens or lengthens. The direction of the pipe does not change while performing this type of grip edit.
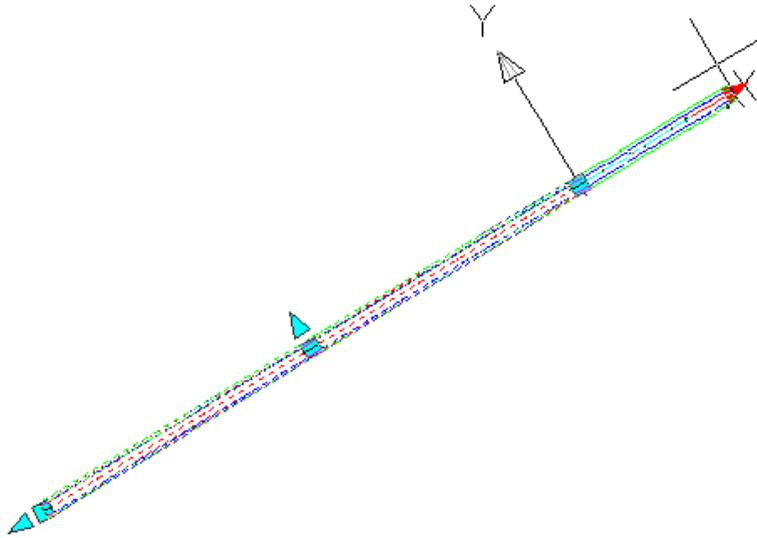
**Midpoint Resize Constraint Grip**

The midpoint resize grip lets you resize the pipe width based on the pipe sizes available in the parts list and the part catalog for the currently selected pipe.

When you use the midpoint resize grip on a pipe, a series of parallel lines display next to the pipe. Each parallel line represents an available pipe width. Tooltip text displays the pipe width. You can snap to any of the parallel lines and select a new pipe width.
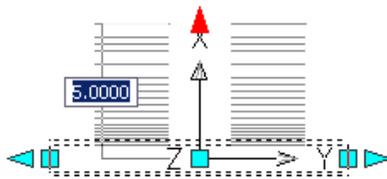


**To use grips to edit a pipe**

1  In the drawing, click a pipe in a pipe network.

   The following illustration shows grips for a pipe in a pipe network:
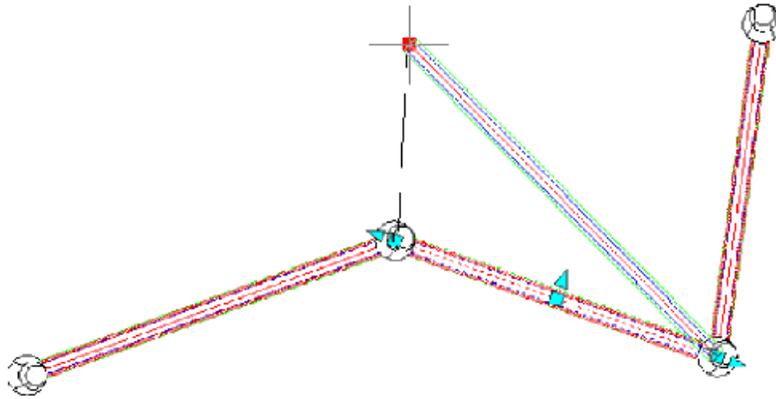
2   Do one of the following:

- Use the end grips to adjust the pipe length.

- Use the center grip to move the location of the pipe.

- Use the midpoint resize grip to change the width of the pipe.

## Editing Pipe Networks Using the Storm Sewers Extension

You can use the Storm Sewers extention to open and edit pipe networks created with AutoCAD Civil.

To open pipe networks in the Storm Sewers Extension, they must first be exported to an *.stm* file.

The *.stm* file can then be opened by the Storm Sewers Extension and edited.

For example, you can create pipe networks in AutoCAD Civil, and then import those pipe networks into Storm Sewers to perform analysis tasks. After performing hydraulic analysis tasks on the pipe network, you can then import that same pipe network back into AutoCAD Civil.

Refer to the following table to find more information about using the Storm Sewers Extension:

| For information about ... | See... |
|---|---|
| exporting AutoCAD Civil pipe networks to an *.stm* file | Exporting Pipe Network Data to an .stm File (page 1279) |
| importing an *.stm* file into the Hydraflow Storm Sewers Extension | the Hydraflow Storm Sewers Extension User's Guide Help |
| editing pipe networks in the Hydraflow Storm Sewers Extension | the Hydraflow Storm Sewers Extension User's Guide Help |
| importing an *.stm* file into AutoCAD Civil | Importing Pipe Network Data from an .stm File (page 1278) |
| adding or editing hydraulic data (HGL, EGL, and so on) | Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280) |

You can also use the EditInStorm Sewers command to browse to an .stm file, and automatically open it in the Storm Sewers Extension.

For more information about using the Storm Sewers Extension to edit pipe networks, see the Storm Sewers Extension Help.

When you are finished making changes to the pipe network in the Storm Sewers Extension, save the project to save the changes to the .stm file.

You can import that .stm file back in to AutoCAD Civil, and overwrite (update) the previous version of that pipe network data, if desired. You also have the option of importing it with a new pipe network name.

**To use the EditInStormSewers command**

1  Do one of the following:

  ■ On the Modify tab, click Pipe Networks to display the Pipe Networks contextual tab. On the Pipe Networks contextual tab, click Analyze panel ➤ Storm Sewers drop-down ➤ Edit In Storm Sewers .

  ■ Click a pipe network in the drawing to display the Pipe Networks contextual tab. On the Pipe Networks contextual tab, click Analyze panel ➤ Storm Sewers drop-down ➤ Edit In Storm Sewers .

  ■ At the command line, enter EditInStormSewers and then press Enter.

2  On the Export To Storm Sewers dialog box, select the pipe networks to export, and click OK.

   **NOTE** You must select one or more entire pipe networks. You cannot select individual parts, or a series of connected parts within a pipe network.

3  Enter a name and select a location for the .stm file, and click Save.

The .stm file is created, and the Storm Sewers Extension launches. The exported pipe network data is displayed in Storm Sewers Extension. For more specific information about using the Storm Sewers Extension to edit pipe networks, see the Storm Sewers Extension Help.

## Quick Reference

Ribbon

Click Pipe Networks tab ➤ Analyze panel ➤ Storm Sewers drop-down ➤ Edit In Storm Sewers 

Command Line

 EditInStormSewers

Dialog Box

 Export to Storm Sewers Dialog Box (page 2078)

# Grip Editing Structures

When you move a structure, all pipe ends connected to it move along with the structure.

Elevations of the original pipe ends connected to the structure are typically maintained. However, part rules control elevations and slope of connected pipes under certain conditions. See Part Rules (page 1247) for more information.

It is also important to understand the automatic resize behavior that is built into junction structures. This automatic resize behavior is designed to ensure that appropriate aspects of a junction structure's geometry (such as elevations of connected pipes, sump depths, and vertical pipe clearance areas) are maintained even

when the junction structure's elevation changes. For more information, see Junction Structure Resize Behavior (page 1211).

**Rim and Sump Depth Grips**

While viewing a structure in a profile view, use the structure's rim or sump depth grip to adjust the rim or sump depth. In the following illustration, the rim grip is located at the top of the structure. The sump depth grip is located just above the bottom of the structure.
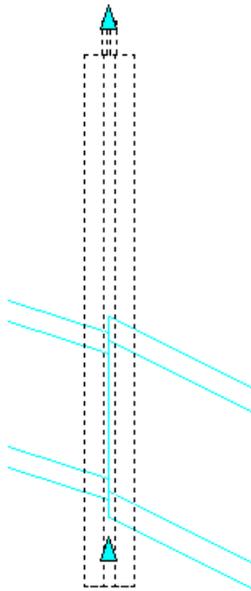


What the structure grips change depends on how the structure's Insertion (Rim) and Sump Behavior properties are set. For example, if you have the structure's Automatic Surface Adjustment property enabled, grip editing the structure modifies the surface adjustment factor. If you have the Automatic Surface Adjustment property disabled, grip editing the structure modifies the rim (insertion) elevation. For sump behavior properties, if you have the Sump Depth property enabled, grip editing the structure modifies the sump depth. If you have the Sump Elevation property enabled, grip editing the structure modifies the sump elevation. See the Part Properties tab (page 2052) of the Structure Properties dialog box for more information.

**NOTE** You cannot grip edit a structure in section view.

**To use grips to edit a structure**

■ To move the location of a structure that has pipes connected to it, in the drawing, click the center grip of the structure and move it to a new location.

■ While viewing structures in a profile view, use the structure's rim or sump depth grips to adjust the elevation of either.

## Editing Pipes Connected to Structures

Use the Connected Pipes tab on the Structure Properties dialog box to quickly edit pipes connected to a single structure.

When you have pipes connected to a structure, you may need to adjust the values of properties such as elevation, and or pipe diameter/width. To do this, you can use the Connected Pipes tab on the Structure Properties dialog box to edit the properties of multiple pipes connected to the same structure.

Using this tab, you can

■ edit inner diameter, inner width, or inner height of pipes connected to a structure

■ quickly edit elevations on multiple pipes to match the elevation of a selected pipe

**To edit pipes connected to a structure**

1　In the drawing, right-click the structure you want to edit, and then select Structure Properties from the shortcut menu.

2　Click the Connected Pipes tab.

3　To edit the following properties, click the value in the cell and enter a new value, or select one from the list: inner diameter (for circular pipes), inner width or inner height (for non-circular pipes).

> **NOTE** On this tab, the read-only slope value refers to the slope of the pipe as it is coming out of the structure (slope out).

4　To edit the elevation for a single pipe, select the pipe and edit the invert, centerline, or crown elevation value.

5　To edit elevations for multiple pipes, see for Matching Elevations on Connected Pipes (page 1219) more information.

6　Click Apply to make the changes, or click OK to make the changes and close the Structure Properties dialog box.

## Junction Structure Resize Behavior

In AutoCAD Civil, junction structures have built in resize behaviors that control how far you can move the rim, connected pipes, and sump up and down.

For example, if you move the top-most pipe invert up, raising its elevation to the point where it reaches the top of the structure (rim), the rim moves up automatically. This is to ensure that the pipe remains connected to the structure *below* the vertical pipe clearance area. The resize behavior allows the junction structure to maintain the frame height, cone height, and honor the vertical pipe clearance area, at the top of the structure, with no pipes connecting in to that part of the structure.

If you try to move a connected pipe into the structure's vertical pipe clearance area, you will notice in your drawing that the top of the structure (rim) automatically moves up.

This is just one example of the resize behavior that is built in to junction structures. Other resize behaviors control how far you can move the junction structure's rim or sump up and down. These resize behaviors are described in more detail in the following sections.

Before editing junction structures, make sure you understand the following about the resize behavior:

■ Elevation changes to the junction structure's rim, connected pipes, or sump, could occur when the structure is grip edited, or when the referenced surface changes, or when pipe elevations are changed.

■ The resize behavior of junction structures is independent of the behavior supplied by part rules. However, part rules can be configured so that they affect the structure resize behavior. For more information, see Part Rules (page 1247).

■ Note that the resize behaviors explained here apply only to junction structures. They do not apply to inlet/outlet structures or null structure types.

■ To verify that a structure is categorized as a junction structure, check that the Part_Type property is set to Struct_Junction. This is displayed on the Part Properties tab of the Structure Properties dialog box, in the Part Data section.

## Junction Structures With Connected Pipes

Moving pipes that are connected to junction structures invokes certain structure resize behaviors.

This section describes the structure resize behavior that is invoked in the following situations:

■ moving the top of a junction structure (rim) up or down

■ moving a pipe that is connected to a junction structure up or down

■ moving the bottom of a junction structure (sump) up or down

These changes to elevation in the junction structure's rim, connected pipes, or sump, could occur because the structure is grip edited, or because the referenced surface is edited, or because a connected pipe elevation is moved. Any of these types of edits may result in elevation changes to parts of the structure.
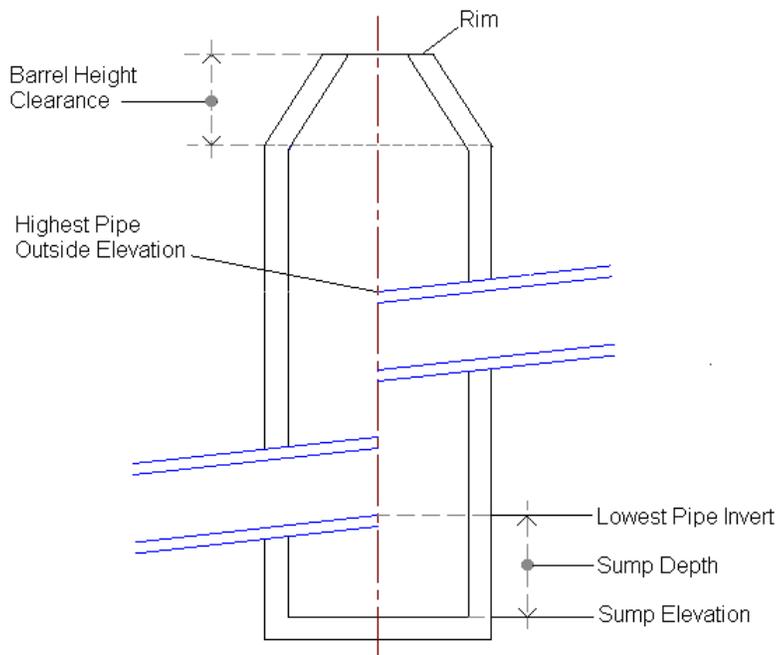


### Moving the Rim

When a junction structure's rim (top of structure) is moved **up**, the rim simply moves to the new elevation. Connected pipes, and sump elevations, remain unchanged.

If you try to move a junction structure's rim **down**, to an elevation where it violates the vertical pipe clearance area, you will not be able do so. The rim must always be located above the vertical pipe clearance area.

### Moving Connected Pipes

The following table describes the resize behavior that occurs when connected pipes are moved **up**.

When a pipe that is connected to a junction structure is moved **up**, to the point where it violates the vertical pipe clearance, the top (rim) of the structure automatically moves up so that the pipe does not touch the vertical pipe clearance area. In this case, the structure's rim elevation changes (moves up).

If you move the pipe **up**, but not to the point where it touches the vertical pipe clearance area, then the pipe is moved up, but the rest of the structure remains where it is. In this case, the structure's rim elevation remains unchanged.

**Understanding the Control Sump By Property**

When you are moving a connected pipe down, there is a part property associated with the junction structure that affects the structure resize behavior in this situation. That property is the Control Sump By property, and it can be set to Depth or Elevation. When this property is set to Depth, the sump depth is controlled by the depth, based on the lowest pipe invert. When this property is set to Elevation, the sump depth is controlled by the elevation value of the sump. The resize behavior is different depending on how this property is set on the structure.

When a connected pipe is moved **down**, to a point where the pipe moves below the connected structure's sump elevation, and the Control Sump By property is set to Depth, since the sump elevation is set to be controlled by the depth, then the sump elevation will be moved down, to a new elevation. This new sump elevation will be the depth of the lowest pipe invert, minus the sump depth value.

If a connected pipe is moved **down**, to a point where the pipe moves below the connected structure's sump elevation, but the Control Sump By property is set to Elevation, then the sump will automatically move down, to an elevation equal to the lowest pipe elevation. This sets the sump depth to zero.

**Moving the Sump**

Using the sump depth grip, you can move a junction structure's sump **up** to the point where it meets the lowest connected pipe. You will be prohibited, however, from moving the sump up past the point where it touches the lowest connected pipe.

When you move a junction structure's sump **down**, the sump will simply move to the new elevation, and not affect any connected pipes, or the structure's rim. This behavior is the same whether the sump is controlled by depth or by elevation.

## Junction Structures With No Connected Pipes

When there are no pipes connected to a junction structure, the rim and vertical pipe clearance take precedence over the sump.

The following sections describe the resize behaviors on junction structures that have no connected pipes.

**Moving the Rim**

When you move the structure's rim **up**, and the Control Sump By property is set to Depth, the sump moves up the same distance as the rim.

When you move the structure's rim **up**, and the Control Sump By property is set to Elevation, the sump moves up the same distance as the rim.

When you move the structure's rim **down**, the sump does not move. You can move the rim down, but only to a location just above the sump. The rim must always be above the sump.

**Moving the Sump**

When you move the structure's sump **up**, and the Control Sump By property is set to Depth, the new sump elevation can be below, at, or above the vertical pipe clearance area. This does not affect the rim elevation, and it sets the sump depth to zero.

When you move the structure's sump **up**, and the Control Sump By property is set to Elevation, the sump can only move up as far as the bottom of the vertical pipe clearance area.

When you move the structure's sump **down**, the sump is moved to its new location (new sump depth), but the rim elevation does not change.

## Adding a Part to a Pipe Network

Add a part (pipe or structure) to an existing pipe network.

To add one or more parts within an existing pipe network, using the Network Layout Tools toolbar, select the type of parts you want to insert (pipes and structures, pipes only, or structures only), and then specify insertion points in the drawing.

You can also add parts to a pipe network by copying and pasting a part within an existing pipe network.

### To add a part to a pipe network

1   In the Prospector tree, click the pipe network object, right-click, and click Edit Network. The Network Layout Tools toolbar is displayed.

2   On tNetwork Layout Tools toolbar, click ![icon] and select one of the following choices: Pipes and Structures, Pipes Only, or Structures Only.

3   In the drawing, specify the insertion point by clicking on the pipe network object you want to connect to.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Pipe Network drop-down ➤ Pipe Network Creation Tools ![icon]

Menu

Pipes menu ➤ Create Pipe Network By Layout

Command Line

CreateNetwork

Dialog Box

Create Pipe Network (page 2023)

## Swapping Pipe Network Parts

Select an existing part in a pipe network and replace it with a part that is the same type, but is from a different part family and/or part size.

A part can only be swapped with a part that is the same type.

For example, a pipe can only be swapped with another pipe; a junction structure can only be swapped with another junction structure; an inlet/outlet structure can only be swapped with another inlet/outlet structure.

Note that a null structure can only be swapped with a junction structure, since it connects two pipes together.

If you want to replace a part with a different part type, you must delete the part, and then add the new one.

Note that when swapping parts, connection elevations are maintained. For example, pipe invert elevations remain the same if a pipe is swapped with another pipe, or if a structure with one or more pipes attached to it is swapped with another structure.

You can swap with any part from the pipe network's current parts list.

NOTE Part swapping should not be confused with part sizing. In part sizing, an existing part is resized within the same part family.

**To swap a part in a pipe network**

1 In the drawing, right-click the pipe or structure object to swap and click Swap Part.

2 In the Swap Part Size dialog box, expand a part family. Select the part to swap and click OK.

The new part is displayed in the drawing. View the description of the new part by right-clicking on the part and selecting Pipe Properties.

## Disconnecting Pipe Network Parts

Select a part in a pipe network and disconnect it from the part or parts it is currently connected to.

When you disconnect a part, the disconnected part can then be moved in the drawing without moving parts that were attached to it.

Even though a part may be disconnected from the other parts in the same pipe network, the disconnected part still belongs to the same pipe network.

The following behavior applies to the connectivity of pipes and structures:

■ When grip editing is used to move, lengthen, or shorten a pipe, the connectivity to any structures connected to that pipe will be lost. There is one exception to this rule: if the pipe end being gripped is moved to the location within the structure it was originally attached to, then the connectivity is maintained.

■ When grip editing a pipe that is attached to a structure, moving the pipe does not move the structure. The pipe becomes disconnected from the structure. The only affect this type of edit may have is to possibly resize the structure vertically to accommodate for a new sump elevation or top of barrel when the pipe is disconnected.

■ When grip editing a pipe, if no 3D OSNAP is used (any value other than zero), the elevation of the end of the pipe is maintained when the pipe is grip edited and connected to a new structure.

■ When grip editing a pipe that is moved over an existing pipe that it will "break," the end of the pipe being edited moves to the elevation of the pipe being broken.

**To disconnect a part in a pipe network**

■ In the drawing, right-click the pipe or structure object and click Disconnect From Part. If you selected a structure, you will be prompted to select the pipe to disconnect from.

The part name is displayed in the Pipe Networks collection on the Prospector tab.

## Connecting Pipe Network Parts

Select a disconnected part in a pipe network, or add a new part, and connect it to other pipe network parts.

When you use this command, the disconnected part automatically snaps to connect to the selected part.

**To connect a disconnected part in a pipe network**

1  In the drawing, right-click the disconnected pipe or structure object and click Connect To Part.

2  Click the part you want to connect to.

The second part automatically snaps to connect to the first part.

## Deleting Pipe Network Parts

Delete one or more parts from a pipe network.

When you delete a pipe or structure from a pipe network, the object is erased from the drawing, and also removed from the Prospector list view.

You can delete pipes or structures using the Basic Modify Tools ➤ Erase command, or using the Delete Pipe Network Object button on the Network Layout Tools toolbar, or by selecting the object in the Prospector list view and choosing Delete from the shortcut menu.

**To delete a pipe or structure using the Basic Modify Tools**

■  In the drawing, right-click the pipe or structure you want to delete. Choose Basic Modify Tools ➤ Erase, or select the part and press the Delete key on your keyboard.

The structure or pipe object is erased from the drawing and removed from the Prospector list view.

**To delete a pipe or structure using the Network Layout Tools**

1  In Toolspace, on the Prospector tab, expand the Pipe Networks collection, then expand the Networks collection.

2  Right-click the pipe network and choose Edit Network to display the Network Layout Tools toolbar.

3  On the Network Layout Tools toolbar, click 🗙 Delete Pipe Network object.

4  In the drawing, select the pipe or structure you want to delete.

The structure or pipe object is erased from the drawing and removed from the Prospector list view.

**To delete a pipe or structure in the Prospector List View**

1  In Toolspace, on the Prospector tab, expand the Pipe Networks collection, then expand the desired pipe network to display the Pipes and Structures collections beneath it.

2  If you want to delete one or more pipe objects, click the Pipes collection in the Prospector tree. If you want to delete one or more structures, click the Structures collection. Hold down the Shift key to select multiple objects.

3  Right-click and click Delete from the shortcut menu.

The structure or pipe object is erased from the drawing and removed from the Prospector list view.

## Deleting a Pipe Network

Delete an entire pipe network, erasing it from the drawing and removing it from the Pipe Networks collection in the Prospector tree.

When you delete a pipe network, all parts belonging to the pipe network (pipes, structures, and null structures) are also deleted.

**To delete a pipe network**

1 In Toolspace, on the Prospector tab, expand the Pipe Networks collection, then expand the Networks collection.

2 Right-click the pipe network and click Delete.

The pipe network, and all its associated child objects, are erased from the drawing and removed from the Pipe Networks collection in the Prospector tree.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Pipe Networks ➤ Right-click <network-name> ➤ Delete

## Copying a Pipe Network

Create a copy of an existing pipe network.

You can copy an existing pipe network, copying the parent pipe network object and all of its associated pipes and structures, and paste this to a new location in the drawing.

When a pipe network is copied, a naming convention is used to name the new pipe network object. For example if the original pipe network object is named 'Pipe Network 1', the new pipe network object is named 'Pipe Network 1 (1)' in the Prospector tree. If Pipe Network 1 is copied again, the new pipe network object is named 'Pipe Network 1 (2)'. If 'Pipe Network 1 (1)' is copied, the new pipe network object is named 'Pipe Network 1 (1) (2)'.

**To copy a pipe network**

1 In the drawing, select the entire pipe network.

> **NOTE** Make sure you select all of the pipe network. If you do not copy all of the parts associated with the pipe network, the selected parts will be copied to the existing pipe network.

2 Use any of the following methods to copy the pipe network to the clipboard: CTRL+C, the AutoCAD COPYCLIP command, or Clipboard ➤ Copy from the right-click shortcut menu.

3 Press CTRL+V.

At the command line, you are prompted to specify an insertion point for the pipe network.

4 Click an insertion point in the drawing area.

The pipe network is pasted in the drawing and added as a new pipe network to the Pipe Networks collection in the Prospector tree.

5 To edit the name of the pipe network, right-click the pipe network in the Prospector tree. Click Network Properties.

The Pipe Network Properties dialog box is displayed with the Information tab (page 2033) active. The Name field contains the name of the copied pipe network, which is, by default, " <pipe network-name> (1)."

6 Enter a new name in the Name field.

7 Optionally, enter a new description of the pipe network in the Description field.

8 Click Apply to make the changes, or click OK to make the changes and close the Pipe Network Properties dialog box.

# Renaming Pipe Network Parts

You can easily rename some or all parts in a pipe network.

In order to manage and easily identify pipe runs, you may need to name pipes and structures according to naming conventions that make sense for your projects.

It is easy to rename a single pipe network object, but what is the best way to rename multiple or all parts within a pipe network? If you need to update just one or a few parts within a pipe network, you may want to use the Pipe Network Vistas (page 2064) in the Prospector, or just right-click on the part in the drawing and enter a new name for the object on the Pipe Properties or Structure Properties dialog box.

If you want to rename many parts within a pipe network, use the RenamePipeNetworkParts command. This command lets you rename one, all, or a sequence of connected parts within a pipe network.

Using this command, you can also select a name template to apply the new part names.

**To rename connected parts in a pipe network**

1 Select the pipe network. Click Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Rename Parts 🖾 .

2 In the drawing, select the first part (pipe or structure) within the pipe network that you want to rename.

   **NOTE** If you select a pipe or structure that is not connected to another pipe or structure, you can only rename that single part.

3 Select the last part in the same pipe network that you want to rename.

4 On the Rename Pipe Network Parts dialog box (page 2077), you may enter or choose a new name (or name template) for the selected pipe network parts. You can choose to rename just the selected structures from the sequence of consecutive parts selected, just the pipes, or both.

5 You can enter a new starting number or leave this field blank.

6 Select one of the following Name Conflict Options. These determine how to handle any conflicts with the naming or numbering, if there are any conflicts.

   ■ Skip Number: When this option is selected, if a name and number combination is already being used by a pipe network part in the drawing, then the number is incremented until that name\number combination is available as a name for the selected part.

   ■ Rename Existing Parts: When this option is selected, if a name and number combination is already being used by a pipe network part in the drawing, then that part will be renamed so that the selected part can use that desired name instead. The original part will be renamed using the next name and number combination that is available.

7 Click OK. The parts are renamed.

To view the updated part names, do one of the following:

■ In the drawing, right-click on a part and select Pipe Properties or Structure Properties. View the updated pipe or structure names on the Information tab of the Pipe Properties or Structure Properties dialog box.

■ In the Prospector tree, expand the Pipe Networks ➤ Networks collection, and then expand the pipe network containing the renamed parts. Click Pipes or Structures and view the updated part names in the Pipe Network Vistas (page 2064).

### Quick Reference

Ribbon

Select the pipe network. Click Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Rename Parts 🗐

Menu

Click Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Rename Parts 🗐 .

Command Line

RenamePipeNetworkParts

Dialog Box

Rename Pipe Network Parts (page 2077)

## Matching Elevations on Connected Pipes

When multiple pipes enter and exit a structure, you may need to set pipe in and pipe out elevations to consistent, matching values.

This feature lets you quickly set pipe elevations in cases where multiple pipes connect to a single structure. This provides a method for quickly and accurately matching multiple pipe elevations to the invert, crown, or centerline elevation of a selected pipe.

You can also add a drop value that lets you adjust that elevation value between incoming and outgoing pipes.

**To match elevations on pipes connected to a structure**

1  In the drawing, right-click the structure, select Structure Properties, and then click the Connected Pipes tab.

2  Press and hold the CTRL key on your keyboard and select more than one of the pipes displaying on the Connected Pipes tab.

   **NOTE** You must select multiple pipes for the match crown, centerline, and invert options to display on the right-click shortcut menu.

3  While still holding down the CTRL key, right-click and select one of the following options from the shortcut menu.
   - Match crowns
   - Match centerlines
   - Match inverts

4  On the Match Elevation dialog box, click the pipe you want to match crown, centerline, or invert elevations to.

5  Optionally, enter a Drop Amount to adjust the elevation by the specified value.

6  Click OK to make the changes and close the Match Elevation dialog box.

7  On the Connected Pipes tab, verify that the selected elevation value has been edited.

8  Click Apply to make the changes, or click OK to make the changes and close the Structure Properties dialog box.

# Displaying Pipe Networks

You can display pipe networks in plan, profile, and section views.

## Displaying Pipe Networks in Profile Views

You can display selected pipe network parts, or entire pipe networks, in profile views.

To display pipe networks in a profile view, you must have valid station offset data (from the profile view alignment) for the pipe networks.

### Editing Parts in Profile View

If you make changes to the pipe network in plan view, such as moving, swapping, deleting, or resizing parts, those changes are reflected when the parts are displayed in profile view.

**NOTE** You cannot move the X, Y locations of parts in a profile view. However, you can grip edit elevations, edit object properties, or resize parts.

### Editing Display Styles in Profile View

Initially, the object style comes from the part, but it can be overridden in profile view. To edit display characteristics for pipe network parts displayed in a profile view, you right-click the profile view, click Profile View Properties, and then click the Pipe Networks tab. For example, you can select the parts you want to be drawn or not drawn in the profile view, or change layers or object styles. When you edit these display characteristics on the Profile View Properties tab, they would only affect how the object is displayed in profile view. For more information, see the Pipe Networks tab (page 2184) on the Profile View Properties dialog box.

### Pipe Slopes in Profile View

The calculated slope of pipes in a profile view may or may not reflect the apparent slope of the actual linework drawn in the profile view. This is because the slope of the pipe is the slope of the actual model from end to end. If a pipe does not geometrically correspond to the alignment defining the profile view, the pipe shown in that profile view will appear slightly different. Also, the length of the pipe in a profile view may not reflect the true length of the pipe as displayed in plan view. This is because the displayed length of the pipe in the profile may be different than the true length of the pipe, as displayed during layout.

### Curved Pipes in Profile View

Curved pipes represented in a profile view are always represented as a straight line from the start point of the pipe to the end point of the pipe. In reality, when a pipe is projected onto a profile, and depending on the combination of horizontal and vertical geometry, the pipe would have some vertical deflections here and there, even when the pipe is actually straight with no change in grade along its length. This is especially true for curved pipes. Therefore, it is important to understand that AutoCAD Civil represents all pipes in profile views as straight lines drawn end to end.

### Pipe Crossings in Profile View

A pipe crossing in a profile view refers to the location where a pipe crosses an alignment or sample line and is displayed in its profile view. When displaying a cross section of a pipe network in AutoCAD Civil, locations of intersected pipes (pipe crossings) are represented at the correct elevation. However, the projected pipe may not show the same elevation at that location for the reasons described in the previous paragraph. Typically, there is no need to represent pipe networks as both projected and crossings in the same profile. For more information, see Displaying Pipe Crossings (page 1222).

**Structures in Profile View**

Pipe network structures will only display in a profile view if the insertion point of the structure is within the limits of the profile view. Therefore, when the structure's insertion location is outside the profile view, the structure does not display in the profile view.

In the Structure Style dialog box, Display tab you can activate components to display connected pipes in profile and section views.

Pipes can be represented to indicate the size/elevation where they are connected to the structure. Pipes represent the true inside section of the pipe, and are exaggerated to the profile view scale in vertical.

**NOTE** The pipes are represented at the elevation of the pipe where they meet the structure, which is typically the centerline.

**To represent pipe network parts in a profile view**

1  Make sure that the desired profile view is already created. See Profiles (page 1019) for more information.

2  Make sure that the pipe network parts you want to display in the profile view have already been created in plan view, and have valid station offset data from the alignment they reference.

3  Select the profile view. Click Profile tab ➤ Launch Pad panel ➤ Draw Parts In View  .

4  In the drawing, select the pipe network parts you wish to add to the profile view, or enter E to select an entire pipe network.

5  Select a profile view.

The pipe network parts are displayed in the profile view.

## Quick Reference

Ribbon

Click Profile tab ➤ Launch Pad panel ➤ Draw Parts In View 

Menu

Pipes menu ➤ Edit Pipe Network

Command Line

AddNetworkPartsToProf

# Displaying Pipe Networks in Section Views

You can display pipe network parts in section views.

To display pipe networks parts in a section view, the following data must exist:

■  Pipe network parts drawn in plan view

■  A sample line and section view containing pipe or structure crossings

■  Valid station offset data for the pipe network parts

Only the pipe network parts that cross the sample line are displayed in the section view.

**NOTE** Pipe sections are drawn using the FACETDEV (Facet Deviation) system variable. FACETDEV is set to 0.5 by default which will usually cause circular pipe sections to be displayed with eight segments. If you set FACETDEV lower, to 0.05 for example, the appearance of the pipe sections will become more circular. The lower this variable is set, the more segments will be used to draw the pipe, up to a maximum of 500.

Pipes and structures that cross the sample line are displayed in the section view as a crossing version of the part. This is unlike profile views where pipe network parts can be displayed both as projected and crossing versions. See Displaying Pipe Crossings (page 1222) for more information.

Pipe objects that are displayed in a section view are displayed according to the style specified on the Sections tab. Structures are displayed in a section view as a representation of the body of the part, even if only a portion of the shape crosses the sample line.

When you use the Create Sample Line Group dialog box to create a sample line, you can select any and all pipe networks in the drawing. However, only pipe network parts that actually cross the sample line are displayed in the section view.

If you wish to display pipe network parts in a section view, a Pipe Network Section object gets created when you create a sample line with pipe network parts that cross it. The Pipe Network Sections collection is displayed under the Sample Line Groups collection in the Prospector tree. See Sections (page 1123) for more information.

**NOTE** Network parts displaying in a section view are exaggerated vertically to match the vertical scale factor of the view in which they are being represented.

**To represent pipe network parts in a section view**

1  Make sure the pipe network parts you want to display in the section view have already been created in plan view and have valid station offset data associated with them.

2  Create the desired sample line and section view. See Sections for more information. The procedure is similar to the procedure for creating surface sections.

# Displaying Pipe Crossings

Pipes that cross or intersect a profile view or section view are displayed as pipe crossings.

Pipes that cross or intersect a profile view can be displayed as projected, or as pipe crossings. Pipes that cross or intersect a section view are always displayed as pipe crossings.

When a pipe is displayed as projected in a profile view, it is drawn as straight linework from pipe beginning to pipe end. It is important to understand, however, that when a pipe is displayed in a profile view, depending on the combination of horizontal and vertical geometry, the pipe will have some vertical deflections, even when the pipe is actually straight with no change in grade along its length. This is especially true for curved pipe. This is how AutoCAD Civil displays pipe in profile views.

When plotting a cross section of a pipe network in AutoCAD Civil, the location of an intersected pipe is displayed at the correct elevation, but the same pipe displayed in a profile view may not show the same elevation at that location due to the behavior described above.

The following illustration shows a pipe crossing and a projected pipe in a section view.

Crossing pipe

Projected pipe

14+00.00          15+00.00
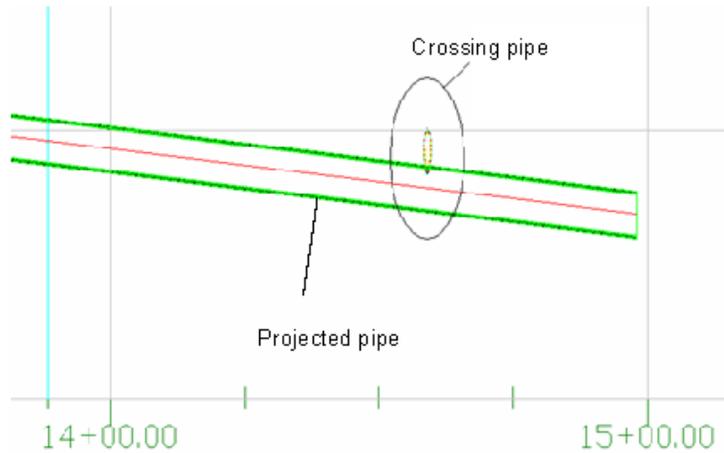
**See also:**

■ [Displaying Pipe Networks in Profile Views](#) (page 1220)

## Pipe End Cleanup Option

You may need to clean up pipe to pipe connections displayed in plan or profile views.

This topic describes how to understand and use the pipe end clean up option that exists on the Plan and Profile tabs of the Pipe Style dialog box.

Using this option, you can define a pipe style that either has this option selected (turned on or in use), or not selected (turned off or not in use).

When this option is checked for plan views, the appearance of pipe to pipe connections in plan view is cleaned up so that they more accurately represent the way pipe to pipe connections should look in plan views.

When this option is checked for profile views, the appearance of pipe to pipe connections in profile view is cleaned up as follows:

■ The overlapping pipe ends are trimmed to the inside edges of the pipes when only the inside edges are displayed, or when both the inside and outside edges of the pipes are displayed.

■ The outside edges of the pipes are trimmed only when the outside edges of the pipes are displayed (when the inside edges of the pipes are not displayed).

The following illustration shows a profile of a pipe to pipe connection with the pipe end clean up option turned on.
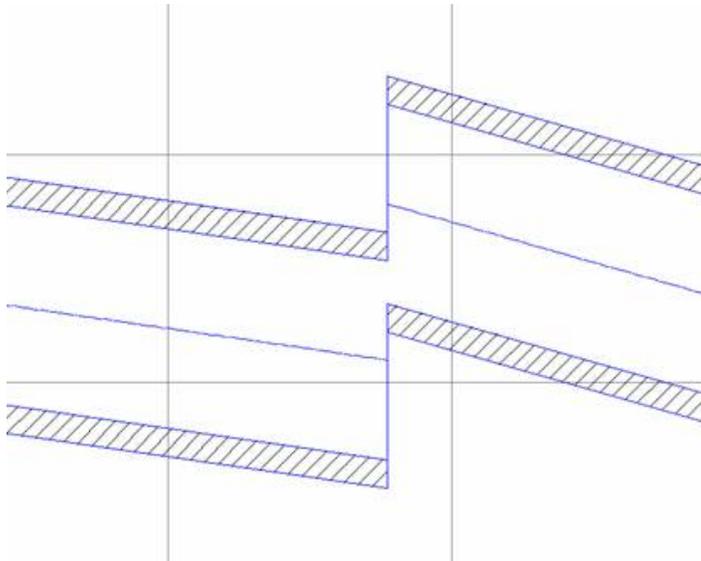
When this option is not checked, the appearance of pipe to pipe connections displayed in profile view is not cleaned up. The following illustration shows a profile of a pipe to pipe connection with the pipe end clean up option turned off.
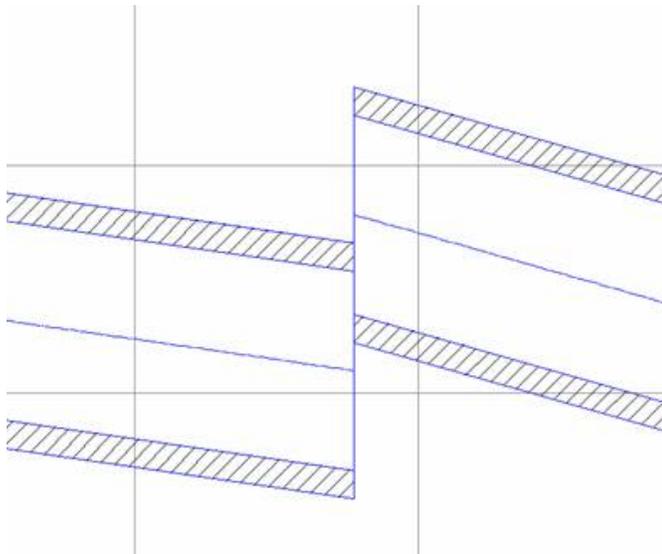


By default, this option is not checked (turned off). But you can simply turn it on for existing (legacy) pipes to have the pipe to pipe connections cleaned up.

It is important to understand the following behavior regarding this option:

■ The pipe end clean up option can only be used when there is a null structure between two pipes.

■ This option behaves the same on straight and curved pipes displayed in profile views.

■ This option trims (removes) the overlapping pipe ends to the inside edges when only the inside edges are displayed, or when both the inside and outside edges are displayed.

■ This option trims to the outside edges only when the outside edges are displayed (when the inside edges are not displayed).

**To clean up the appearance of pipe to pipe connections in profile views**

1   Right-click on a pipe in a pipe to pipe connection and click Edit Pipe Style.

2   Click the Profile tab on the Pipe Style dialog box.

3   At the bottom of this dialog box, place a check mark next to the Clean Up Pipe To Pipe Connections option.

4   Click OK.

In profile views, the pipe ends in pipe to pipe connections are trimmed (cleaned up).

**To clean up the appearance of pipe to pipe connections in plan views**

1   Right-click on a pipe in a pipe to pipe connection and click Edit Pipe Style.

2   Click the Plan tab on the Pipe Style dialog box.

3   At the bottom of this dialog box, place a check mark next to the Clean Up Pipe To Pipe Connections option.

4   Click OK.

In plan views, the pipe ends in pipe to pipe connections are trimmed (cleaned up).

# Changing Flow Direction

You can change the flow direction on consecutive parts in a pipe network.

Because some pipe network rules rely on the flow direction assigned to pipes, this command lets you assign direction as needed so that rules can be applied properly.This feature also allows for the display of appropriate flow direction arrows in labels.

The Change Flow Direction command is also useful when other applications using AutoCAD Civil pipe network model data need to be able to determine flow direction within a network.

You may need to change the flow direction of an entire pipe network, or only a portion of a pipe network. Using the Change Flow Direction command, you are prompted to select a path of consecutive parts in a pipe network.

To change the flow in a series of connected parts, select a path by clicking on a starting part, and an ending part within a pipe run.

Changing flow direction can only be performed on a single pipe network at a time. Although you can select structures as well as pipes when using this command, the feature affects pipes only.

**To change pipe network flow direction**

1   Select the pipe network. Click Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Change Flow Direction ✍ .

    A prompt is displayed indicating that you should select the part (pipe or structure) that you want to be used as the upstream starting point in plan view.

2   In the drawing, click a pipe or structure in the pipe network to define the upstream starting point for the flow direction change.

    A prompt is displayed indicating that you should select the part (pipe or structure) that you want to be used as the downstream ending point in plan view.

**3** Do one of the following:

- Select a path for the flow direction change by clicking a pipe or structure to define the downstream ending point. You can continue clicking consecutive parts in a pipe network to define the path for the change flow direction.

- You can click Undo at any time during this command to undo the previously selected part and path.

**4** When you have finished defining the path for the flow direction change, press Enter to end the command and implement the flow direction change on the parts in the selected path.

A prompt indicating that the flow direction has been successfully applied to the part(s) is displayed.

The Flow Direction property value on the Part Properties tab of the Pipe Properties dialog box is changed (from Start to End to End to Start, or from End to Start from Start to End).

### Quick Reference

Ribbon

Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Change Flow Direction ⌖

Menu

Pipes menu ➤ Change Flow Direction

Command Line

ChangeFlowDirection

# Checking for Interferences

Interference checking lets you quickly identify pipe network parts that may be in conflict with each other.

This feature compares the actual 3D model of parts to check for interferences. You can run an interference check to identify pipe network parts that physically overlap, collide, or intersect in an inappropriate way, or that have violated predefined, proximity-based criteria.

For example, you may want to run an interference check on one or more pipe networks to see if any parts physically overlap or are too close to one another. You can select style-based visual markers to identify the interferences, or you can choose to display the interferences as true, three-dimensional representations. You may choose to leave the interference conditions as they are, or you may decide to resolve them by moving parts in the drawing.

When you run an interference check, you can choose a single pipe network or you can choose two pipe networks. When you run an interference check on a single pipe network, it check for interferences on parts included in that network. When you choose to check two pipe networks, it checks for interferences between the two pipe networks.

The following illustration shows two different pipe networks with structures that physically overlap. Running an interference check would quickly indicate that these parts interference with one another.
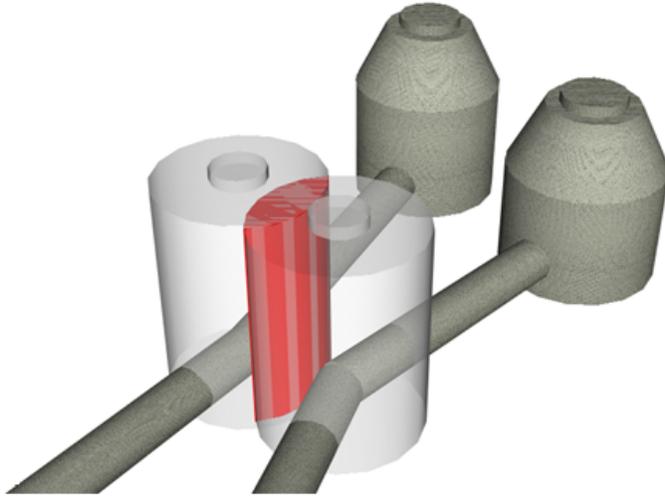
Another type of interference that can be detected is when two pipes intersect without a structure joining them, as shown in the following illustrations:
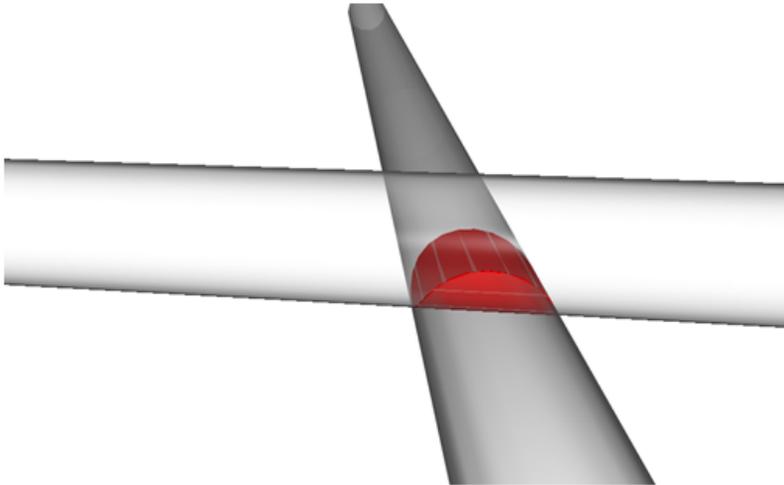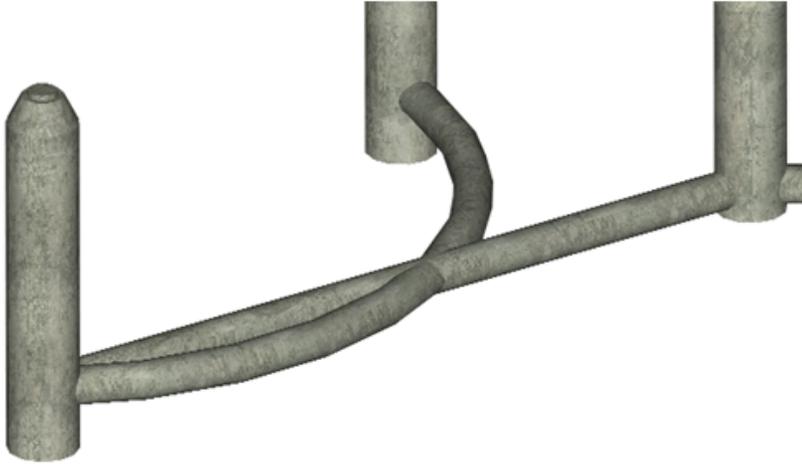
If desired, you can also perform a check for pipe network parts in their 3D representation that are too close to one another, according to predefined distance or scale factor criteria. In the following illustration, the two pipe networks shown are less than ten feet away from each other at the closest point, even though they do not physically collide:
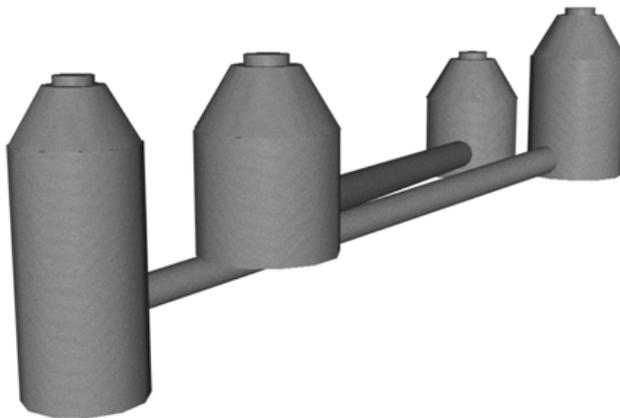


You can run an interference check that identifies parts in their 3D representation that are less than a specified distance away from any part in the other network. The distance is checked in every direction surrounding each part.

When you run an interference check, an Interference Check object gets created and is displayed in the Prospector tree in the Pipe Networks ➤ Interference Checks collection. For more information, see Viewing Interference Check Results (page 1232).

## Proximity Checking Criteria

When you run an interference check, you can enable or disable 3D proximity checking criteria.

If you enable the 3D proximity checking option on an interference check, you can identify pipe network parts that are too close to one another in their 3D representation, according to a specified distance or a scale factor. You can check a single pipe network or two pipe networks.

When you use proximity checking on a single pipe network, interferences are created for parts that are less than a specified distance from one another. When you use this feature to check two pipe networks, an interference is created for any parts from one network that are less than a specified distance or scale factor from any parts in the second network.

You can use the 3D proximity check feature to ensure that the distance between two different types of utility networks meets a certain requirement. For example, a sewer and water crossing typically requires that the water line be a minimum of 18 inches above the sewer line. Running an interference check with the Use Distance option set to 18 inches identifies any parts that were less than 18 inches away from the other pipe network.

The following illustration shows how the distance and or the scale factor 3D proximity checking value is used to determine proximity interference violations. The darker shaded outlines represent the actual parts in the selected pipe network. The lighter shaded parts represent the area that is used to identify interferences.
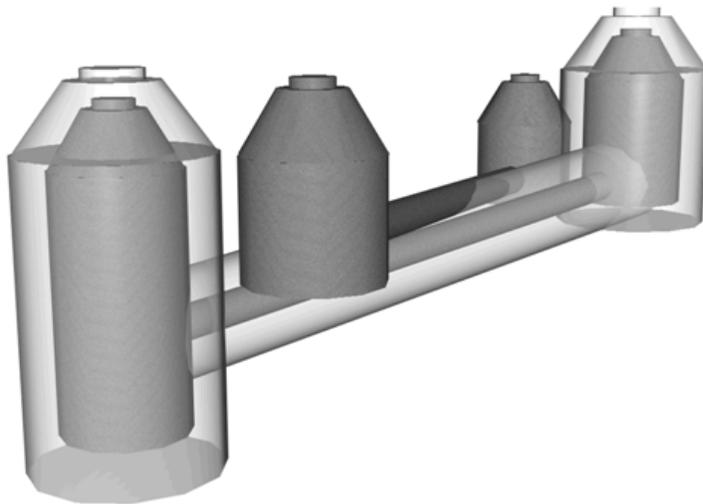


**Specifying a Distance**

When you choose to enter a specified distance to perform the proximity check, you can enter a value that is used to perform the proximity check. When you run the check, interferences are created for any pipe network part that is less than the specified distance away from any other part in the specified network(s), in any direction.

**Specifying a Scale Factor**

When you use the Use Scale Factor option is selected, you can enter a value that is used as a scale factor. When you enter a scale factor value, AutoCAD Civil calculates the limits of the part (the pipe or the structure) multiplied by the scale factor value that is specified. For example, if the pipe diameter is 600mm, and you specify a scale factor value of 2, AutoCAD Civil identifies any interferences within a distance of 1200mm (or the pipe diameter, which is 600, multiplied by the scale factor value, which is two). When the interference check is run, any pipe network parts within that distance away from each other (in any direction) are marked as an interference.

**To define 3D proximity check criteria**

1   In the Create Interference Check dialog box, click 3D Proximity Check Criteria.

**2** In the Criteria (page 2071) dialog box, click Apply 3D Proximity Check to enable this feature.

**3** Do one of the following:

■ Click Use Distance to enter a distance. When the interference check is run, interferences are created for any pipe network part that is less than the specified distance away from any other part in the specified network(s).

■ Click Use Scale Factor to enter a scale factor. When the interference check is run, any pipe network parts within that distance away from each other (on any side) are marked as an interference.

**4** Click OK in the Criteria dialog box.

**5** Click OK in the Create Interference Check dialog box to run the interference check.

The following prompt is displayed:

Interference check created. Number of interferences found: X.

**6** Click OK at this prompt.

To view the results, in Toolspace, on the Prospector tab, expand the Interference Checks collection, and then select the interference check. The Interference Check data is displayed in the Panorama window and Toolspace item view. See Viewing Interference Check Results (page 1232) for more information.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Pipe Networks ➤ Interference Checks ➤ Create Interference Check

Dialog Box

Criteria (page 2071)

## Running an Interference Check

Run an interference check to identify parts in a pipe network that collide or violate proximity criteria.

You can check for interferences on a single pipe network, or you can check two pipe networks to determine if parts collide or violate proximity criteria.

**To run an interference check on a single pipe network**

**1** Click Analyze tab ➤ Design panel ➤ Interference Check ⬚ .

**2** In the drawing, click any part in the pipe network.

**3** Click again on the same part in the pipe network or on any other part in the same pipe network.

**4** In the Create Interference Check dialog box, in the Name field, specify a name for this interference check, or accept the default.

**5** Optionally, enter a description in the Description field.

**6** Specify the pipe network(s) for this interference check using the Network 1 and Network 2 fields.

These should be the same pipe network(s) that you clicked in step 3. If desired, you may change the pipe network(s) here.

To run an interference check on a single pipe network, make sure the same pipe network name is displayed in the Network 1 and Network 2 fields.

**7** Specify the layer the interference check results will be displayed on.

**8** Specify an interference style and a render material.

**9** If you want the interference check to identify pipe network parts that are too close to one another, using a specified distance or scale factor, click the 3D Proximity Check Criteria button.

**10** To use proximity checking options, in the Criteria dialog box, click Apply 3D Proximity Check, and then do one of the following:

- Click Use Distance and enter a distance. When the interference check is run, interferences are created for any pipe network parts that are less than the specified distance away from each other (on any side).

- Click Use Scale Factor and enter a scale factor. When the interference check is run, interferences are created for any pipe network parts that are less than the specified scale factor distance away from each other (on any side).

**11** Click OK in the Criteria dialog box.

**12** Click OK in the Create Interference Check dialog box to run the interference check.

The following prompt is displayed:

Interference check created.

Number of interferences found: X

**13** Click OK at this prompt.

To view the results, in Toolspace, on the Prospector tab, expand the Interference Checks collection, and then select the interference check. The Interference Check data is displayed in the Panorama window and Toolspace item view. See Viewing Interference Check Results (page 1232) for more information.

**To run an interference check between two pipe networks**

**1** Click Analyze tab ➤ Design panel ➤ Interference Check 🐾 .

**2** In the drawing, click any part in the first pipe network.

**3** Click any part in the second pipe network.

**4** In the Create Interference Check dialog box, in the Name field, specify a name for this interference check or accept the default.

**5** Optionally, enter a description in the Description field.

The Network 1 and Network 2 fields display the names of the pipe networks previously selected. If desired, you can use these fields to change the pipe networks for this interference check.

**6** Specify the layer the interference check results will be displayed on.

**7** Specify an interference style and a render material.

**8** If you want the interference check to identify pipe network parts that are too close to one another, using a specified distance or scale factor, click 3D Proximity Check Criteria.

**9** To use proximity checking options, click Apply 3D Proximity Check on the Criteria dialog box, and then do one of the following:

- Click Use Distance to enter a distance. When the interference check is run, interferences are created for any pipe network parts that are less than the specified distance away from each other (on any side).

■ Click Use Scale Factor to enter a scale factor. When the interference check is run, interferences are created for any pipe network parts that are less than the specified scale factor distance away from each other (on any side).

**10** Click OK in the Criteria dialog box.

**11** Click OK in the Create Interference Check dialog box to run the interference check.

The following prompt is displayed:

Interference check created.

Number of interferences found: X

**12** Click OK at this prompt.

To view the results, in Toolspace, on the Prospector tab, expand the Interference Checks collection, and then select the interference check. The Interference Check data is displayed in the Panorama window and Toolspace item view. See Viewing Interference Check Results (page 1232) for more information.

### Quick Reference

Ribbon

Analyze tab ➤ Design panel ➤ Interference Check 

Toolspace Shortcut Menu

Prospector tab: Pipe Networks ➤ Interference Checks ➤ Create Interference Check

Dialog Box

Create Interference Check (page 2070)

## Viewing Interference Check Results

After running an interference check, view the results in the drawing or by looking at the data in the Toolspace list view.

In the drawing, interferences are displayed according to the choices defined in the Interference and Render Material options specified on the Create Interference Check dialog box. These choices are also displayed on the Information tab of the Interference Check Properties dialog box.

To view interference data in the list view, in Toolspace, on the Prospector tab, expand the Pipe Networks ➤ Interference Checks collection. Click on the Interference Check object in the Prospector tree. The interference data is displayed in the list view. You can right-click on the status column and Zoom To an interference in the drawing. If you choose Delete, it deletes only the interference, not the part. In this view, you can also right-click an interference and select Properties to display the Interference Properties dialog box.

## Updating an Interference Check

Update an interference check after pipe network parts have been changed.

An interference check is marked as out of date ▽ when any pipe network part in the interference check is changed. Even changing a description on a pipe network part that is not in interference causes an interference check to become out of date. So does moving, deleting, or adding parts to any network included in the interference check.

**NOTE** Out-of-date icons are displayed only if the Toolspace drawing item modifier icon ▽ display is active. For more information, see Drawing Item Modifier Icons (page 141).

When you update an interference check, you can leave the interference checking conditions as they are currently set, or you can change them before rerunning the check.

Use the Interference Check Properties dialog to change interference check options, criteria, and other properties.

**To update an interference check**

1 In Toolspace, on the Prospector tab, expand the Pipe Networks ➤ Interference Checks collection.

2 Right-click the interference check and click Rerun Interference Check.

The following prompt is displayed:

Interference check created.

Number of interferences found: X

3 Click OK.

The interference check data is updated in the drawing and in the Prospector tree.

**To change interference check conditions or properties**

1 In Toolspace, on the Prospector tab, expand the Pipe Networks ➤ Interference Checks collection.

2 Right-click the interference check and click Interference Check Properties.

3 Change information about the Interference Check Properties (page 2071) dialog box and click Apply to save the changes.

4 Click OK to accept all changes and rerun the interference check.

The following prompt is displayed:

Interference check created.

Number of interferences found: X

5 Click OK.

The interference check data is updated in the drawing and in the Prospector tree.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Pipe Networks ➤ Interference Checks ➤ Create Interference Check

Dialog Box

Criteria (page 2071)

# Setting Interference Styles

You can create styles to display interference conditions in your drawing.

Like other AutoCAD Civil style features, you can set color, size, layer and other styles for interferences.

You can create different interference styles to use for various types of interferences. For example, you can create an interference check and an interference style to specifically identify parts that are too close to one another (but not overlapping), displaying these parts in a certain color.

Use the Pipe Network ➤ Interference Style collection on the Settings tab to create and manage interference styles.

**To create interference styles**

1 In Toolspace, on the Settings tab, expand the Pipe Network collection, then expand the Interference Styles collection, and click New.

The Interference Style (page 2074) dialog box is displayed.

2 Click the Information tab, enter a name for the interference style and, optionally, a description.

3 Click the View Options tab (page 2074) and set interference style options for symbols (markers) and solids that represent interferences.

4 Click the Display tab (page 2075) and set the display style options.

5 Click OK to save changes and close this dialog.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Pipe Network ➤ Interference Styles ➤ New

Dialog Box

Interference Style (page 2074)

# Deleting Interference Checks

You can easily delete interference checks.

Once you have finalized your drawing or pipe network layout, you may want to delete interference checks from the Prospector tree. When you delete an interference check, it is removed from the Prospector tree. Any visual markers displayed in the drawing to indicate interferences are also removed.

**To delete an interference check**

1 In Toolspace, on the Prospector tab, expand the Pipe Networks ➤ Interference Checks collection.

2 Right-click the interference check and click Delete.

A verification prompt asks if you are sure you want to delete the interference check.

3 Click Yes.

The interference check object is deleted from the Prospector tree. Any visual markers for the interference checking displaying in the drawing are also deleted.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Pipe Networks ➤ Interference Checks ➤ Delete

# Part Properties

Use the Pipe Properties dialog box and the Structure Properties dialog box to view or edit properties associated with any given pipe network part.

Part properties define a variety of characteristics of the part, including the part family the part belongs to, a part type, and properties defining the part's size, shape, elevation, automatic resizing behaviors, and other behaviors. Part properties originate from definitions of the part in the part catalog.

After you insert a part into a drawing, you can edit many of its properties. Those changes apply only to the currently selected part in the drawing. For example, once you add a structure, such as a manhole, to a drawing, you can change the structure's inside diameter by editing the value for the Inner Structure Diameter property, in the Part Data collection, on the Part Properties tab. The default value for the structure's Inner Structure Diameter property that is maintained in the part list (or the part catalog) is not affected.

Part properties are organized into the following categories:

| Category | Description |
| --- | --- |
| General | These properties specify the surface and alignment referenced by the part. For pipes, there are properties that specify flow direction. For structures, the structure elevation at its insertion point is specified. |
| Geometry | For pipes, these properties specify general characteristics of the pipe, such as the flow direction method, flow direction, and the surface and alignment referenced. For structures, these properties specify characteristics such as the structure's rotation angle, offset, station data, and the number of pipes that are currently connected to the structure. |
| Resize Behavior | The resize behavior property is specific to pipes. It specifies the behavior that occurs when the pipe is automatically resized. You can choose to have the pipe hold to its crown, invert, or centerline elevation when it is resized. |
| Hydraulic Properties | For both pipes and structures, these properties specify hydraulic information, such as hydraulic grade lines and energy grade lines, the pipe network may have received from the Storm Sewers Extension. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280). |
| Insertion (Rim) and Sump Behavior | These properties are associated with junction structures only. They specify the structure's insertion point (structure rim) and sump elevation and adjustment behavior. |
| Part Data | For both pipes and structures, these properties specify a variety of part characteristics, such as part type, part subtype, and the part size name as defined in the part catalog. Some of these |

| Category | Description |
|---|---|
| | properties define the part's basic primitive shape and behavior, and therefore are read-only. |

The Pipe Properties and the Structure Properties dialog boxes have the following tabs:

■ **Information**. Specifies the name, description, object style, and object render material assigned to the pipe or structure object.

■ **Part Properties**. Displays a list of properties that exist for this part. These properties define the basic size and shape of the part, as well as the part family the part belongs to. This section also specifies hydraulic properties for the pipe network, if they exist. You can edit values for certain properties on this tab.

■ **Rules**. Displays a list of the rules that are associated with the part. These rules originate from the default rule of the part in the part catalog. You can edit the part's rules once you have inserted the part into a drawing. Rules are used to validate certain properties of the part, and also to determine elevation during layout, and during certain editing commands.

The Structure Properties dialog box has the following additional tab for viewing data about pipes connected to the structure:

■ **Connected Pipes**. Displays a list of each pipe connected to the structure, and property values for each pipe. You can edit values for certain properties, such as elevation data, on this tab. It's very useful for quickly editing elevations, slopes, diameters or descriptions for multiple pipes connected to a structure.

**See also:**

■ Part Rules (page 1247)

■ Pipe Properties Dialog Box (page 2039)

■ Structure Properties Dialog Box (page 2051)

**NOTE** Changes made to either the Pipe Properties or the Structure Properties dialog box affect only the selected part in the drawing. It does not affect the part definition in the part catalog or in a parts list.

**To edit pipe or structure properties**

1 Right-click a pipe or structure in the drawing. Click either Pipe Properties or Structure Properties.

2 In either the Pipe Properties or the Structure Properties dialog box, click the Part Properties tab (page 2040).

3 Click a property value, edit it, and click Apply or OK to save changes.

For example, to change the inner diameter of a structure, scroll to the Part Data property group, click the value for the Inner Structure Diameter property, and select a new value.

4 Click Apply or OK to save changes.

For more information, see Part Properties (page 1235).

**NOTE** Changes made to either the Pipe Properties or the Structure Properties dialog box affect only the selected part in the drawing. It does not affect the part definition in the part catalog or in a parts list.

**To edit pipe or structure rules**

1 Right-click a pipe or structure in the drawing. Click either Pipe Properties or Structure Properties.

**2** In either the Pipe Properties or the Structure Properties dialog box, click the Rules tab (page 2043).

**3** Click the desired property value in the Value column, enter a new value, and press Enter.

**4** Click Apply or OK to save changes.

For more information, see Part Rules (page 1247).

### Quick Reference

Command Line

　EditPipeProperties or EditStructureProperties

Dialog Box

　Pipe Properties Dialog Box (page 2039)

　Structure Properties Dialog Box (page 2051)

# Part Catalog and Parts Lists

The pipe networks feature references a part catalog and a parts list that define the size, shape, and certain behavior of the parts (pipes and structures) you insert into drawings.

### Part Catalog

The AutoCAD Civil part catalog contains definitions for all the pipe network parts that you can insert into a drawing. It is organized at the top level into two basic domains — one for pipe parts, such as cylindrical or rectangular pipes, and another for structure parts, such as headwalls, catch basins, and so on.

Part catalog content is further organized into types, subtypes (also referred to as shapes), and part families. The following table illustrates how the default part catalogs are organized for pipe parts and structure parts.

| Part Domain | Part Type | Part Shape | Part Family |
|---|---|---|---|
| Pipes | Pipe | Circular | Concrete<br>Ductile Iron<br>PVC |
| | | Egg-Shaped | Concrete |
| | | Elliptical | Concrete<br>Concrete Horizontal |
| | | Rectangular | Concrete Box |
| Structures | Inlet-Outlet | Rectangular | Concrete Variable Height<br>Concrete |
| | Junction with Frames | Rectangular | Slab Top Circular Frame<br>Slab Top Rectangular Frame<br>Two-Tier Circular Frame |
| | | Cylindrical | Slab Top Circular Frame<br>Concentric<br>Eccentric<br>Eccentric Two-Tier Circular Frame |

| Part Domain | Part Type | Part Shape | Part Family |
|---|---|---|---|
| | Junction without Frames | Rectangular | Rectangular |
| | | Cylindrical | Concentric Cylindrical |

The files that make up the part catalog content are typically installed to this location: *C:\Documents and Settings\All Users\Application Data\AutoCAD\Civil2010\enu\Pipes Catalog*.

**NOTE** The pipe network part catalog is not related to the corridor (subassembly) catalogs that are available through the Content Browser. Also note that pipe network parts cannot be accessed through a Tool Palette.

### Parts List

A parts list contains a set of pipe network parts – pipes and structures – that you can use in a pipe network. You have parts lists so that you do not have to navigate through the entire part catalog looking for the specific pipe network part that you need.

When you open a new blank drawing by selecting a AutoCAD Civil drawing template, you will typically find default parts lists available, such as Standard. You can create your own parts lists to suit the needs of each project. For example, you may want to create a parts list that contains only the types of parts you will use in a particular project or pipe network. This saves you from having to navigate through the entire part catalog to select a part. While there is only one pipe network part catalog included with AutoCAD Civil, you can create multiple parts lists for your projects.

A parts list also serves as a way to store defaults, such as styles, render materials and design criteria (rules) to any given part upon creation. Parts lists are objects that you can access through the Pipe Network Properties dialog box, the Pipe Network Layout Tools toolbar, and the Toolspace Settings tree.

**NOTE** You can also create a Full Parts list that contains all of the parts that exist in the current metric or imperial part catalog. For more information , see Creating a Full Parts List (page 1239).

## Creating a Parts List

You can create a parts list that contains only the parts (pipes and structures) you will need on a specific pipe network, drawing, or project.

A part list is a subset of available parts in a catalog. It provides quick access to just the parts you need for a specific project.

You can create a parts list by copying an existing parts list and editing it, by creating a new parts list from scratch, or by dragging and dropping a parts list from one drawing into another drawing.

**To create a parts list**

1 In Toolspace, on the Settings tab, expand the Pipe Networks collection, right-click Parts List, and click New. The Network Parts List (page 2035) dialog box is displayed.

2 Click the Information tab, enter a name for the parts list and, optionally, a description.

3 Click the Pipes tab to add pipe parts to this parts list, or the Structures tab to add structure parts.

4 On either the Pipes tab or the Structures tab, right-click the parts list name at the top of the Name field and click Add Part Family. The Part Catalog dialog box (page 2037) dialog box is displayed.

5 In the part catalog, click the check box next to the part families you want to add, and click OK.

6 Click Apply on the Pipes tab or the Structures tabs to save the changes.

**7** On either the Pipes tab or the Structures tab, right-click a part family and click Add Part Size. The Part Size Creator dialog box is displayed.

**8** To add individual sizes of the part, click on and highlight the rows you want to add.

**NOTE** For many part definitions, optional properties are available to the part. For more information, see Assigning Optional Properties to a Part Size (page 1242)

**9** To add all available sizes of the part, place the check mark in the Add All Sizes check box. For more information about adding sizes, see the Part Size Creator (page 2062) dialog box.

**10** Click OK on the Part Size Creator dialog box.

**11** Click Apply to save changes on the Network Parts List dialog box.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Pipe Networks ➤ Parts List ➤ New

Dialog Box

Network Parts List (page 2035)

## Creating a Full Parts List

You can create a parts list that contains all of the part types and associated sizes that exist in the currently selected metric or imperial part catalog.

It may be useful for you to create this master parts list (full catalog) if you are planning to import pipe networks from LandXML or from the Storm Sewers Extension.

Once the full parts list is created, that is the parts list that you would use when importing pipe network data.

During import, using a full a parts list that contains all of the parts that are available in the catalog can sometimes be the best option for ensuring that appropriate part types and sizes will be found when you import pipe network data.

After creating a full parts list, you may want to configure the Storm Sewers migration settings to always use this full parts list during import. For more information, see Setting the Default Parts List For Migration (page 1277).

**NOTE** It is recommended that you do not use the full parts list while manually laying out (creating by layout) new pipe networks or while editing existing pipe networks. Because the full parts list contains several thousand parts, navigating for parts using this large volume of available part types could be time consuming. The CreateFullPartsList feature is intended primarily to support pipe network data import.

**To create a full parts list**

**1** On the Home tab, click Toolspace ➤ Settings tab, expand the Pipe Network collection, and then expand the Parts Lists collection. Note the names of the existing parts lists displayed in the Settings tree.

**2** Do one of the following:

■ From the ribbon, click the Modify tab ➤ Pipe Network ➤ Network Tools panel ➤ Parts List drop-down menu ➤ Create Full Parts List 

■ At the command line, enter CreateNetworkPartsListFull and then press Enter.

The new full parts list is created. By default, this parts list is named Full Catalog. You may want to rename this to suit your needs.

To configure the Storm Sewers migration settings to always use this full parts list during import, see Setting the Default Parts List For Migration (page 1277).

### Quick Reference

Ribbon ➤ Modify Tab

Modify ➤ Pipe Network ➤ Pipe Networks Contextual Tab ➤ Network Tools panel ➤ Parts List ➤ Create Full Parts List ▣

Ribbon ➤ Pipe Networks Contextual Tab

Pipe Networks tab ➤ Network Tools panel ➤ Parts List ➤ Create Full Parts List ▣

Command Line

CreateNetworkPartsListFull

## Copying a Parts List

You can copy an existing parts list in your drawing and give it a new name, or you can copy a parts list from one drawing to another.

**To copy a parts list in a drawing**

1 In Toolspace, on the Settings tab, expand the Pipe Network collection, then expand the Parts Lists collection.

2 Right-click on the parts list you want to copy and click Copy. The Network Parts List (page 2035) dialog box is displayed. A copy of the parts list is created, with Copy of appended to the parts list name.

3 Click the Information tab, change the name for the parts list and, optionally, enter or change the description.

4 Click Apply to save changes.

5 Click the Pipes tab to add or remove pipe parts to this parts list, or the Structures tab to add or remove structure parts.

6 On either the Pipes tab or the Structures tab, right-click the parts list name at the top of the Name field and click Add Part Family. The Part Catalog dialog box (page 2037) dialog box is displayed.

7 In the part catalog, click the check box next to the part families you want to add, and click OK. Note that only part families that are not already included in the current parts list are displayed in the part catalog so that you cannot add the same part family twice in one parts list.

8 Click Apply on the Pipes tab or the Structures tabs to save the changes.

9 On either the Pipes tab or the Structures tab, right-click a part family and click Add Part Size. The Part Size Creator dialog box is displayed.

10 To add individual sizes of the part, click on and highlight the rows you want to add.

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**NOTE** For many part definitions, optional properties are appended to the part name. For more information, see Assigning Optional Properties to a Part Size (page 1242)

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**11** To add all available sizes of the part, place the check mark in the Add All Sizes check box. For more information about adding sizes, see the Part Size Creator dialog box (page 2062).

**12** Click OK on the Part Size Creator dialog box.

**13** Click Apply to save changes on the Network Parts List dialog box.

**To copy a parts list from one drawing to another**

**1** With two drawings open, in the first drawing, in Toolspace, on the Setting tab, expand the Pipe Networks collection, and then expand the Parts Lists collection.

**2** Select the desired parts list, drag it (click and hold) to anywhere in the drawing area of the second drawing, and then release the mouse cursor (drop).

**3** In the second drawing, click the Settings tab, expand the Pipe Network collection, and then expand the Parts Lists collection to see the new parts list.

# Adding a Part Family to a Parts List

Add a part family from the pipe network part catalog to a parts list.

After adding a part family, you must select desired part sizes. If you add a part family but do not select the part sizes, part family content is not displayed in the parts list.

**To add a part family to a parts list**

**1** In Toolspace, on the Settings tab, expand the Pipe Network collection, and then expand the Parts Lists collection.

**2** Right-click the desired parts list and click Edit Parts List. The Network Parts List dialog box is displayed.

**3** Click the Pipes tab to add a pipe part family to this parts list, or the Structures tab to add a structure part family.

**4** On either the Pipes or the Structures tab, in the Name column, expand the parts list to view the part families that are currently included.

**5** Right-click the name of the parts list at the top level and click Add Part Family. The Part Catalog (page 2037) dialog box is displayed.

> **NOTE** Part families that are included in the current parts list are not displayed in the part catalog. Only part families that are available to be added to the current parts list are displayed in the part catalog.

**6** In the part catalog, select the part families you want to add, and click OK.

**7** Click OK or Apply on either the Pipes tab or the Structures tab to save the changes.

After you have added one or more part families to the parts list, the next step is to select the part sizes you want. See Adding Part Sizes to a Parts List (page 1242).

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Pipe Network ➤ Parts Lists ➤ <parts list name>➤ Edit Parts List

Dialog Box

Network Parts List (page 2035)

# Adding Part Sizes to a Parts List

After a part family is added to a parts list, select the part sizes you want to be available in that part family.

When creating new parts lists, you first add the part families, and then add the part sizes that you want to be available within that family.

For example, for the Elliptical Concrete Pipe family, you can add only the part size of 8-inch inner pipe width, or you can click Add All Sizes to add all of the size choices that are available in the drop-down list for the inner pipe width value (for example, 8, 12, 18, etc.).

If you do not add part sizes within a part family, no content is displayed for that part family in the parts list.

For existing parts lists, you can add part sizes at any time.

**To add part sizes to a parts list**

1   In Toolspace, on the Settings tab, expand the Pipe Network collection, and then expand the Parts Lists collection.

2   Right-click the desired parts list and click Edit Parts List. The Network Parts List dialog box is displayed.

3   Click the Pipes tab to add sizes for pipe parts, or the Structures tab to add sizes for structures.

4   Expand the parts list so that you can see the part families that are currently included.

5   Right-click a part family and click Add Part Size. The Part Size Creator dialog box is displayed.

6   Use the Part Size Creator (page 2062) dialog box to add part sizes to the current part family.

    You can select existing sizes one at a time, or you can use the Add All Sizes check box. For example, for the Elliptical Concrete Pipe part family, you can add only the part size of "8-inch inner pipe width", or you can check Add All Sizes to add all the size choices that are available in the drop-down list simultaneously: for example, 8, 12, 18, and 24.

7   For each part size that is added, you may also add optional properties which are not defined in the part family.

    For example, to specify a material for a pipe, add the optional Material property and enter the desired text to set the property. For more information, see Assigning Optional Properties to a Part Size (page 1242).

8   Click OK on the Part Size Creator dialog box.

9   Click OK or Apply on either the Pipes tab or the Structures tab to save the changes.

# Assigning Optional Properties to a Part Size

To add further definition to a part, such as a name for a material, you can assign optional properties to a part size definition in a parts list.

After an optional property is assigned to the part, the property may be viewed or edited in the Part Properties tab (page 2040) of either the Pipe Properties or the Structure Properties dialog box, viewed or edited in the Pipe Network Vistas (page 2064), or it can be displayed as labeling text for the part in the drawing.

An example of an optional property is the Material property available for the Concrete Pipe part family. You can assign this optional property to part sizes within the Concrete Pipe part family.

**To add an optional property to a part size**

1   In Toolspace, on the Settings tab, expand the Pipe Network collection, and then expand the Parts Lists collection.

**2** Right-click the desired parts list and click Edit Parts List. The Network Parts List dialog box is displayed.

**3** Click the Pipes tab to assign an optional property to a pipe part size, or the Structures tab to assign an optional property to a structure part size.

**4** On either the Pipes tab or the Structures tab, in the Name column, expand the parts list so that you can see the part families that are currently included.

**5** Right-click the desired part family and select Add Part Size.

**6** The Part Size Creator Dialog Box (page 2062) is displayed. The optional properties are identified in the Source field.

**7** If the optional property can be assigned, you can click in the Value field and set the property from a list of choices. If the optional property has a static value, no choices are available.

**8** Click in the Value field to select and assign the property.

**9** Click OK on the Part Size Creator dialog box.

**10** Type in the value or string.

**11** Click OK or Apply on the Pipes or Structures tabs to save the changes.

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**NOTE** All optional properties available for a part family can be set, even after the part has been created, by displaying the Pipe Properties or Structure Properties dialog box.

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**To add a user-defined property to a network part**

**1** Navigate to the Part Parameter Configuration xml file (AeccPartParamCfg.xml). The default location is *C:\Documents and Settings\All Users\Application Data\Autodesk\AutoCAD <version>\enu\Pipes Catalog\Aecc Shared Content*.

**2** Open the file and select View ➤ Source in order to modify the file.

**3** Navigate to the <AeccParamDeclaration> section. A sample optional property is the Hazen Williams Coefficient flow analysis. The entry for this is: <AeccDfParameter name="ACHW" desc="Hazen Williams Coefficient" context="FlowAnalysis_HazenWilliams" index="0" datatype="Float" usage="Double_General" unit="" visible="True" internal="True"/>.

**4** Copy an existing optional property and make changes as appropriate to create a new property.

**5** In the <AeccParamUsage> section, the corresponding entry for the sample in step 3 is <AeccOptParam context="FlowAnalysis_HazenWilliams"/>. Copy and modify an entry as appropriate to create an new entry for your property.

**6** Save and close the xml file.

# Setting the Pipe Network Part Catalog

You can configure AutoCAD Civil to access the imperial or the metric pipe network part catalog, or to access specific folders for pipe and structure parts content.

The part catalog contains the definitions for all the pipe network parts that you can insert into a drawing.

The default part catalog location is *C:\Documents and Settings\All Users\Application Data\AutoCAD\Civil2010\enu\Pipes Catalogs*.

**To set the pipe network part catalog**

**1** Click Home tab ➤ Create Design panel ➤ ▼ ➤ Set Pipe Network Catalog 🔧 .

**2** On the Pipe Network Catalog Settings dialog box, click the folder icon to browse to the location of the pipe network catalog folder. This is the top-level folder containing all the pipe network part catalogs. For example, this folder contains subfolders for imperial and metric pipes and structures.

The default part catalog location is *C:\Documents and Settings\All Users\Application Data\AutoCAD\Civil2010\enu\Pipes Catalogs*.

**3** Select the desired folder and click Open.

**4** You may want to specify different pipe and structure catalogs, if they are available.

**5** Click OK to save changes to this dialog box.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ ▼ ➤ Set Pipe Network Catalog

Menu

Pipes menu ➤ Set Pipe Network Catalog

Command Line

SetNetworkCatalog

Dialog Box

Pipe Network Catalog Settings (page 2063)

## Editing Part Family Sizes

Use the Part Builder feature to change the part sizes that are available in a part family.

If a part family does not have the sizes you need, you can modify the part family sizes using the Part Builder feature.

**To edit part sizes in a part family**

**1** Enter PartBuilder at the command line and press Enter.

**2** Select Pipe or Structure from the Part Catalog drop-down list.

**3** Using the catalog tree, locate and select the Part Family you want to edit.

**NOTE** Pipe and structure families are organized by type. For example, pipe families are organized into types such as circular pipes, egg-shaped pipes, and so on. Structure families are organized into inlet-outlet structures, junction structures with frames, and so on.

**4** Click the Modify Part Sizes button , or double-click the Part Family, to display the Part Family browser window.

**5** If this is a Part Family that shipped with AutoCAD Civil, click the Save Part Family As button in the Part Family browser window toolbar

**6** In the Save Part Family As dialog box, enter a new Part Name and Part Description and then click OK. This creates your own version of the selected Part Family you can edit.

**7** In the Part Family browser window, right-click the Size Parameters node, and click Edit Values.
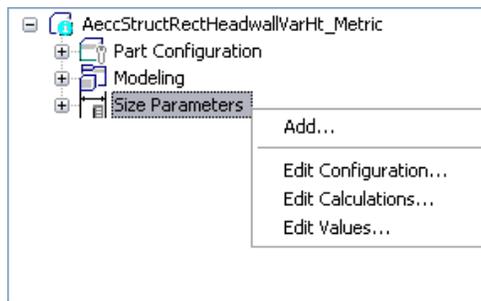
**8** In the Edit Part Sizes dialog box, find the parameter (property) value(s) you want to edit.
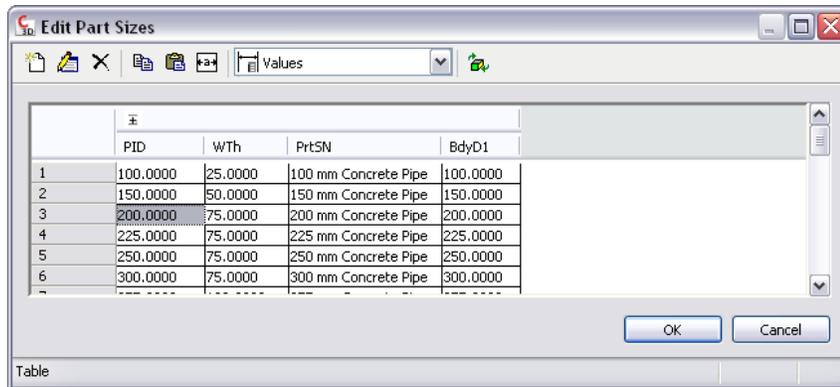


**9** Edit the values as described below:

**Pipe values**

■ To change values for pipe sizes, select the row and size cell. Double-click to edit.

■ To delete a size row, select the row and click the Delete button ✖ on the Edit Part Sizes dialog box.

■ To add a size row, select the closest size and click the New button on the Edit Part Sizes dialog box, and then edit the values in the new row.

**Structure values**

■ To edit list values for structure sizes, select the cell and click the New button on the Edit Part Sizes dialog box.

**10** After you have completed editing sizes, click OK on the Edit Part Sizes dialog box.

**11** In the Part Family browser window toolbar, click the Save Part Family button .

**12** When finished editing part sizes, close the Part Builder by clicking the close button in the upper right corner of the Part Family browser window.

**13** You will be prompted to save the part family with the following dialog box. Click Yes at this prompt.

**14** Next, you may be prompted to save the part family drawing file with the following dialog box. Click No at this prompt.

The new part family and the new part sizes you create will be available through the Network Layout Tools toolbar.

# Sample Part Family Size Editing

This sample exercise walks you through how to edit the part sizes that are available in a part family.

In this example, you will edit the Pipe Inner Diameter (PID) and Wall Thickness (Wth) values in the Concrete Pipe part family.

**To edit part sizes in the Concrete Pipe part family**

1   Enter PartBuilder at the command line and press Enter.

2   Click the Part Catalog drop-down list and select Pipe.

3   In the catalog tree, expand the Circular Pipes node, then select the Concrete Pipe part family.

4   Double-click the Concrete Pipe part family or click the Modify Part Sizes button ▣ .

5   Click the Save Part Family As button ▣ .

6   In the Save Part Family As dialog box, enter a new Part Name and Part Description for this new part family and then click OK. This creates your own version of the selected Part Family you can edit.

7   In the Part Family browser, right-click the Size Parameters node, and click Edit Values.
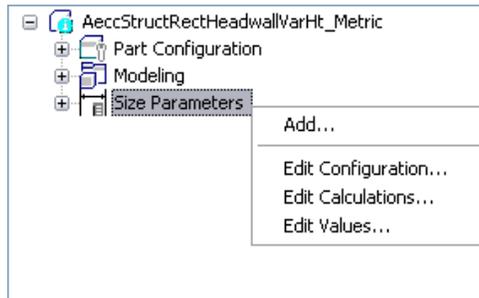


8   In the Edit Part Sizes dialog box, click the Autosize Column Text button then find the Pipe Inner Diameter (PID) value and the Wall Thickness (WTh) values.

9   Select any row in the table and click the New button ▣ on the Edit Part Sizes dialog box. The selected row is copied to the table.

10   In the new table row, edit the PID and WTh values by double-clicking on them and typing in the new values.

11   After you have completed editing sizes, click OK on the Edit Part Sizes dialog box.

12   In the Part Family browser window toolbar, click the Save Part Family button ▣ .

13   When finished editing part sizes, close the Part Builder by clicking the close ▣ button in the upper right corner of the Part Family browser window.

14   You will be prompted to save the part family. Click Yes at this prompt.

15   Next, you may be prompted to save the part family drawing file. Click No at this prompt.

   The new part family will not be available when creating or editing a pipe network until you edit the parts list and add this new part family to your parts list. For more information, see Adding a Part Family to a Parts List (page 1241)

Once you have added the new part family to your parts list, the new part family and the new part sizes you created will be available through the Network Layout Tools toolbar.

# Viewing Part Catalog Content

Use an Internet Explorer window to view content from a published part catalog.

The Internet Explorer view shows an image of the 2D or 3D shape and lists all the defined values associated with the part family. For example, all size parameters, whether defined as table, list, range, and or calculated values (formulas), can be viewed in a read-only mode. Constant values, such as the part domain, type, subtype, part name, description, and the ID associated with the part, are all displayed.
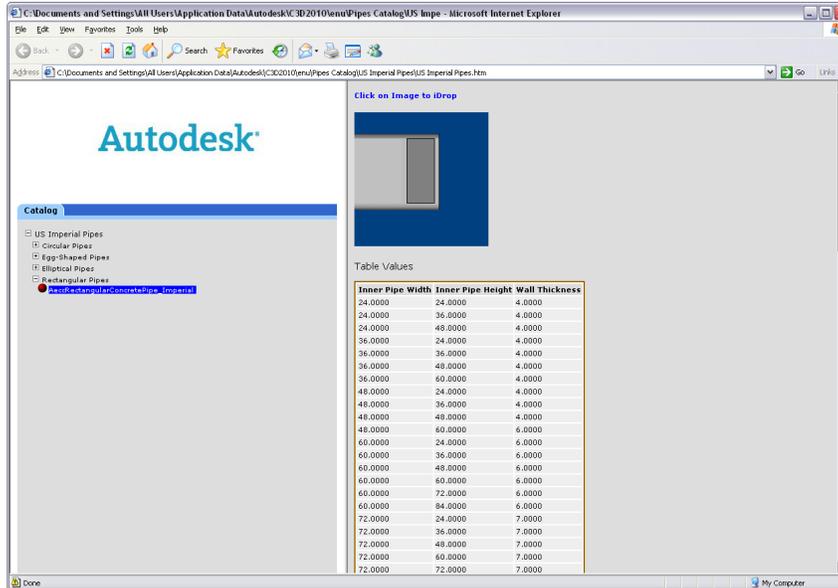


**To view published part catalog content using Internet Explorer**

1  Using Internet Explorer, navigate to the location of the part catalog, for example, *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Pipes Catalog\US Imperial Structures*.

2  Double-click the name of the desired part catalog, for example, US Imperial Structures.htm.

3  In the left pane catalog tree, expand a node and then select a part family.

   For example, expand the Junction Structures With Frames node, and then select the AeccStructEccentricCylinder_Imperial part family.

Content for the selected part family is displayed in the right pane.

# Part Rules

Part rules are properties that affect how pipe network parts behave on creation and when you move or edit them.

AutoCAD Civil pipe network objects use part rules in the following ways:

■  To determine elevations for pipes and structures when they are created.

■  To determine how pipes connect to junction structures.

■  To warn that certain criteria is not met while creating or editing pipe networks.

Primarily, part rules automatically determine reasonable elevations for parts as they are created. The specific elevation behavior you see is based on choices you make, such as how you define the minimum slope, minimum cover, and desired drop across a structure. Rules also provide automatic validation of your pipe

network design, such as checking to see if a structure is too small to accommodate a pipe size, if the length of a pipe is greater than desired, or the maximum cover above a pipe is exceeded.

Rules provide an excellent way to give a good starting point for design. They also provide a way to quickly determine if the location of a series of pipes and structures is actually constructible.

Rules are applied to parts automatically as they are created. However, you can reapply rules to existing parts in your drawing as needed. Reapplying rules ensures that unintended changes are not made. For example, if you move a structure that has pipes connected to it, you may want to reapply rules after moving the structure to ensure that the pipes connected to it still have the desired elevations, minimum cover, and do not exceed a maximum length.

The pipe network rules features provides a high level of design automation, while still allowing you to specify the design according to your judgment and site-specific requirements. When rule parameter values are violated on pipe network parts in the drawing, you can quickly review the violations and determine if you need to edit the drawing or simply allow the rule violations.

**NOTE** It is important to note that rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Storm Sewers Extension. For more information about rules, see Part Rules (page 1247).

### Default Rule Behavior

The default rule behavior for pipe network parts is designed to meet the needs of some typical design scenarios. You can lay out a pipe network without editing, removing, or adding part rules, and this behavior may suit your needs. If you want to change the default rule behavior, you can do so by edit part rule values, or by adding or removing rules from a part using Rule Sets. See Editing Part Rules (page 1258) for more information.

The default pipe network rule behavior in AutoCAD Civil is controlled by the following files:

- C3DPipeRules. This Visual Basic script file is similar in design and architecture to the script file that controls AutoCAD Civil subassembly behavior for corridor modeling.

- C3DPipeRules.xml. This file defines the pipe network part rule parameters and rule names.

These files are located in located in the *\data* folder. The *\data* folder installed to *\\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\data* by default.

### Rules and Parts Lists

Each part in a pipe network parts list references a set of pre-defined rules. When a pipe or structure is inserted into a drawing, the rule set is referenced by the part itself. Therefore, changes made to the part in the part catalog, or removing a part from the parts list, will have no effect on a part that has already been inserted into a drawing.

### Rule Sets

A default "rule set" exists for both pipes and structures that are inserted into your drawing. The rule set behaves like a style, except that you can also override values if desired. Rule sets simply provide a way for you to create a set customized set of rules. You can add or exclude certain rules, edit certain rule values, or change the order in which the rules are processed on a part, and you can save all these changes in a named rule set. This gives you the ability to control and customize the rule behavior for a specific pipe network or pipe network part.

**See also:**

- Creating a Rule Set (page 1259)
- Pipe Rule Set dialog box (page 2037)

■ Structure Rule Set dialog box (page 2050)

## Rule Violations

Rule violations inform you when pipe network part parameter values have not been met or exceed certain limits.

Whenever rules are applied to a part, validation checks occur to inform you, for example, if minimum or maximum cover limits have been violated, or if some other condition needs attention.

When warnings occur due to rule violations, they are displayed as warning icons ![icon] in the Pipe Network Vistas (page 2064) (Panorama window or Toolspace item view). Rule violation information is also displayed on the Rules tab (page 2043) of the Pipe or Structure Properties dialog box, in the Status column. Placing the mouse cursor over the warning icon in the Prospector List View displays tooltip text describing the warning (rule violation) condition. This tooltip displays the same rule violation text that is displayed in the Status column on the Rules tab of the Pipe or Structure Properties dialog box.

The following illustration is an example of the Rules tab on a pipe in a drawing. In this example, the Minimum

Cover and Maximum Length properties for this pipe display with descriptive rule violation text ![icon] in the Pipe Value column because they violate the design criteria specified in the Value column. Note the rule violation descriptive text that is displayed in the Status column; for example, "Minimum pipe cover exceeded" and "Exceeds the maximum pipe length by".
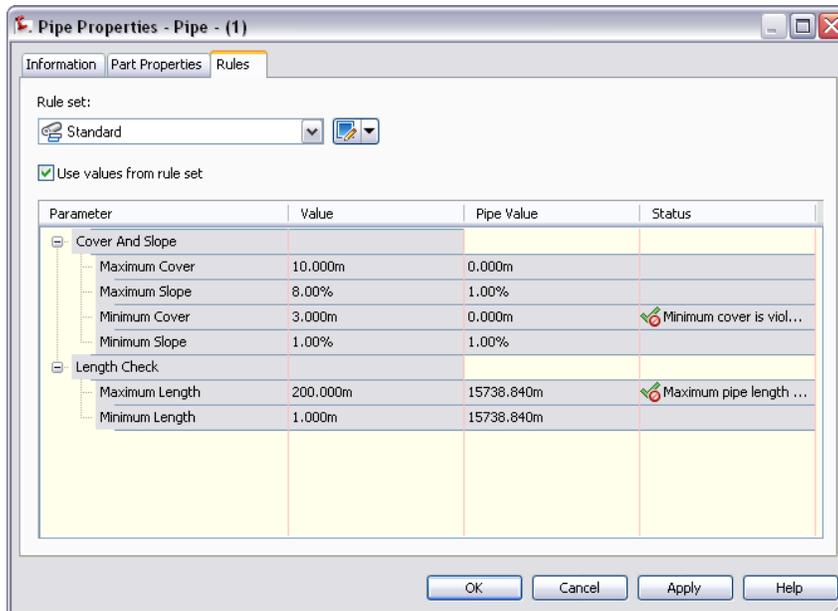


**Fixing Broken Rules**

The pipe network rule feature is a mechanism for visually identifying conditions that violate the design criteria rules that are in place for pipes and structures. You can choose to leave broken rule conditions as they are, or you may desire to resolve the broken rule conditions, depending on the circumstances.

If you wish to resolve a broken rule condition, there are a few different ways you can do this. You can edit the value in the Value column (editing the rule). You can also edit the pipe or structure object itself in the drawing so that it no longer violates the rule. For example, you could move a pipe to a different elevation so that it no longer violates a Minimum Cover rule. In some situations, you may want to reapply rules to a pipe network, or to pipe network parts, when you have a rule violation.

**NOTE** It is important to note that reapplying rules does not automatically resolve rule violations. In some situations, reapplying rules may result in more rule violations, some of which you may decide to simply leave as violations. For example, there may be some situations where you have certain site-specific cover constraints that you want to intentionally violate.

Note that if you edit the Rule Value on a pipe network part in a drawing, it does not affect the rule for other parts in the drawing, for parts in the parts list, or for parts in the parts catalog. It also does not alter any physical characteristics of the part in the drawing. It simply alters the rule limit on that particular part in your drawing.

## Applying Rules to Parts in a Drawing

You can apply rules to pipe network parts to ensure that certain conditions are flagged, and to ensure that parts are placed in correct locations.

For example, you may need to reapply rules to ensure that pipes that have not met the maximum cover value are flagged as rule violations. This enables you to quickly identify parts that may need to be adjusted in your drawing. You may also want to reapply rules after you moved a structure that has pipes connected to it to ensure that the connected pipes still exist in the correct elevation and connection locations.

When you apply rules to a pipe network part, rule values as defined in the Pipe Rule Set or Structure Rule Set for the part are copied to the part.

You may need to reapply rules when:

■ The elevation of a pipe network part has changed due to pipe network parts being moved.

■ A part's rule value has changed.

■ You add or delete a rule from a part.

■ You change the order in which rules are processed on the part.

If you look at the rule set assigned to a part, you can view the rule set, and the rules currently associated with the part. You cannot change the processing order of the rules on the Structure or Pipe Properties Rules tab. However, you can change the order of the rules by displaying the Rules tab on the Rule Set.

Applying rules does not automatically resolve rule violations. You can view rule violations leave them as is, or resolve them.

In some situations, reapplying rules may result in more rule violations, some of which you may decide to simply leave as violations. For example, there may be some situations where you have site-specific cover constraints that you want to intentionally violate.

**To apply rules to a pipe network part**

➤ In the drawing, right-click either a pipe or structure in a pipe network, and then click Apply Rules.

At the command line, a prompt is displayed indicating that rules have been successfully applied to the network part(s).

For more information, see Part Rules (page 1247).

**To apply rules to parts in a pipe network**

1 Click Pipe Networks tab ➤ Modify panel ➤ ▼ ➤ Apply Rules 🖳 .

2 In the drawing, select pipe network parts in an up-slope direction.

3 When finished selecting parts, press Enter to end the command.

At the command line, the following prompt is displayed:

Rules successfully applied to X network parts.

For more information, see Part Rules (page 1247).

# Pipe Rules

These rules primarily govern how the elevations of a pipe object are determined when the object is created. There are also rules to flag pipes that exceed certain values, such as maximum length, for example.

When a pipe rule is violated, you can complete the action, but a warning icon ![icon] is displayed for the pipe object in the Pipe Network Vistas (page 2064) (Panorama window or Toolspace item view), and on the Rules tab (page 2043) of the Pipe Properties dialog box. This indicates that a rule violation exists.

After a pipe object is created, you may make changes to the pipe that result in the pipe rules being violated. For example, you might move a pipe so that the Minimum Cover distance is not met.

The following sections describe the standard rules available for pipe objects.

## Cover And Slope Rule

This rule ensures that a pipe slopes appropriately and warns when a pipe is placed too close to a ground surface.

For gravity-based systems, this is perhaps the most useful rule. It ensures that a pipe slopes in the desired direction under various conditions, while ensuring that minimum cover is not exceeded. In addition, it manages conditions when a pipe drop across structure rule conflicts with the pipe's slope and cover rules.

When drawing a pipe, the pipe attempts to stay within minimum and maximum cover values, unless the minimum or maximum slope is reached. At that point, maximum cover distance is exceeded to satisfy the minimum slope requirement, if it is violated.

This rule ensure the following, with the first pipe having the highest precedence or priority:

■ The connections to structures are at a location as specified by the structure rules, if any are specified.

■ The pipe always slopes in the proper direction, with the minimum slope being honored, unless this is in conflict with a connected structure.

■ The minimum cover is maintained unless this is in conflict with a connected structure.

### Parameters

The following parameters govern the behavior of the pipe Cover And Slope rule.

■ Maximum Cover. Specifies the maximum cover of soil over the length of the pipe, based on the surface being referenced by that pipe. If the maximum cover is exceeded, a rule violation occurs. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

■ Maximum Slope. Specifies the maximum slope of the pipe, expressed in percent. If the pipe slope is greater than the maximum, a rule violation occurs for that object.

■ Minimum Cover. Specifies the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. This is also used to determine the initial elevations of the pipe. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

- Minimum Slope. Specifies the minimum slope of the pipe, expressed in percent. During layout, a pipe will be created according to its minimum slope rule value. If the pipe is edited so that its slope is less than the minimum, you can still edit the pipe as desired, breaking the minimum slope rule, but a rule violation occurs for that object.

## Cover Only Rule

This rule is intended for laying out pipes in a pressure-based pipe network where pipe elevations are determined according to a specified depth below a terrain.

This rule ensures that the minimum cover is met along the length of the pipe, and also validates that both the minimum and maximum cover values are not violated along any length of the pipe.

### Parameters

The following parameters govern the behavior of this rule:

- Maximum Pipe Cover. Specifies the maximum cover of soil over the pipe, based on the surface being referenced by that pipe. If the pipe cover exceeds the maximum cover value, a rule violation occurs for that object. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

- Minimum Pipe Cover. Specifies the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

- Start Cover. Specifies the cover at the start of the pipe.

- End Cover. Specifies the cover at the end of the pipe.

---

**NOTE** For spanning pipe labels **Start Cover** will display the cover at the beginning of the span of pipes that comprise the spanning pipe label. **End Cover** will display the cover for the very end of the span. The start and end of the spanning pipe label depends on the order in which pipes where selected when creating the pipe span.

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## Length Check Rule

This rule governs the behavior that produces a warning condition on a pipe if the pipe length exceeds the value specified for the maximum pipe length, or is less than the value specified for the minimum pipe length.

This rule determines whether the 2D length of the pipe meets the values specified for the minimum and maximum pipe lengths. For this rule, the pipe is measured from pipe end to pipe end, not from the inside edge of the structure.

This rule is useful for design situations that require or recommend a maximum, continuous pipe length. This rule also includes a minimum pipe length warning option. Keep in mind that this rule does not prevent you from exceeding the minimum/maximum lengths. It notifies you that the specified length is being exceeded.

### Parameters

The following parameters govern the behavior of the Length Check rule.

- Maximum Pipe Length. This parameter lets you define a maximum length for pipes. When this parameter (rule) is in use, you can still draw pipes that exceed the defined maximum pipe length. However, the object will be displayed with a warning icon in the Prospector list view. Note that this parameter provides

validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

■ Minimum Pipe Length. This parameter lets you define a minimum length for pipes. When this parameter (rule) is in use, you can still draw pipes that are shorter than the defined minimum pipe length. However, the object will be displayed with a warning icon in the Prospector list view. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is not met.

## Pipe To Pipe Match Rule

This rule governs how a pipe elevation is determined in a pipe network that contains only pipes (no structures), and or when a pipe is inserted onto an existing pipe to break the pipe.

This rule applies only during the following conditions:

■ Creating a pipe network that has pipes only (no structures)

■ Breaking into an existing pipe with another pipe

■ Connecting a pipe to a null structure (pipe to pipe connection with no structure)

■ Connecting a pipe to an existing pipe end

This rule manages conditions so that only continuous runs of pipe are created as typically expected. In pipe networks where the pipe size is constant, pipes connected to other pipes must to match end to end. In pipe networks where pipe sizes change, the point where pipes match may change, depending on the type of system. For example, in pressure systems, it may be typical to match the pipe centerline. In gravity systems, it may be typical to match the crowns of the pipes.

This rule also incorporates a drop value for cases where a pipe-to-pipe connection requires a drop amount.

### Parameters

The following parameters govern the behavior of this rule:

■ Match Location. This parameter controls whether the inserted pipe holds to the pipe's invert, crown, or centerline elevation (location).

■ Drop Value. The drop value allows you to specify an additional drop amount at pipe to pipe connections.

## Structure Rules

These rules govern primarily how a structure object is placed and sized during object creation. They also can determine the elevations of connected pipes. There is also a rule to flag pipes that are too wide to connect to a structure.

Just like pipe rules, when structure rules are violated, a warning icon is displayed for the object in the Pipe Network Vistas (page 2064) (Panorama window or Toolspace item view), and on the Rules tab (page 2057) of the Structure Properties dialog box.

The following sections describe the standard rules associated with structure objects.

## Pipe Drop Across Structure Rule

This rule compares all pipes connected to a single structure and ensures that pipes enter and exit the structure according to the specified drop value.

This rule applies to junction structures only and is intended specifically for gravity-based systems. It is important to note that this rule does not alter the structure in any way, nor does it alter any of the pipes that are connected to the structure. Instead, this rule actually inserts data onto the structure that is read by the pipes when they are using rules. It ensures that the following conditions are achieved when connecting a new pipe to a structure that already has one or more pipes connected to it:

■ A pipe exiting a structure is no higher than the lowest pipe entering the structure.

■ A pipe entering a structure is no lower than the highest pipe exiting a structure.

■ There is always a specified minimum drop distance between the lowest incoming pipe and the highest outgoing pipe.

The drop can be based on a comparison between the crowns, inverts, or centerlines of pipes. A validation check is performed for drops exceeding a certain distance. This determines whether a maximum drop value is violated or whether a drop is required.

**Parameters**

The following parameters govern the behavior of this rule:

■ Drop Reference Location. Specifies the drop location by referencing the pipe's invert, crown, or centerline elevation.

■ Drop Value. Specifies the drop value between the lowest incoming pipe and the highest outgoing pipe connected to the structure.

■ Maximum Drop Value. Specifies what the maximum drop value is between the lowest incoming pipe and any outgoing pipe connected to the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded. This is intended to raise awareness when a drop structure might be needed.

## Pipe Drop Across Structure Rule Examples

The Pipe Drop Across Structure rule applies in a variety of scenarios.

The following examples show how the Pipe Drop Across Structure rule is applied when a new incoming or outgoing pipe is added to a structure that already has pipes connected to it.

The following illustration shows a junction structure that has two incoming pipes already connected to it on the left side of the structure. When a new incoming pipe is added to the right side of the structure, the pipe cover and slope rule determines the optimal connection elevation for the new incoming pipe.

**Example 1: Existing Incoming Pipes with New Incoming Pipe**
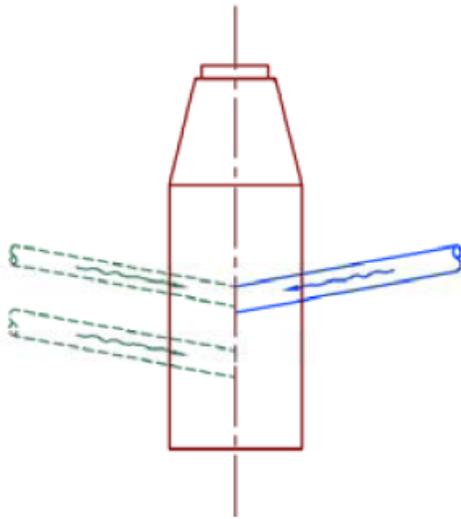


In the following illustration, the junction structure has two incoming pipes already connected to it on the left side of the structure. A new outgoing pipe is added to the right side of the structure. The pipe cover and slope rule determines the optimal connection elevation for the new outgoing pipe.

**Example 2: Existing Incoming Pipes with New Outgoing Pipe**
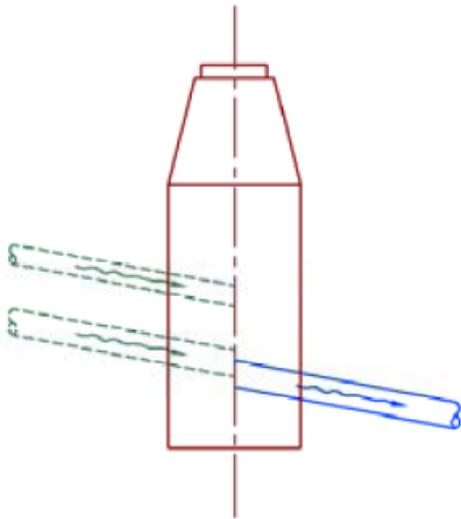


The following illustration shows a junction structure that has two outgoing pipes already connected to it on the left side of the structure. A new incoming pipe is added to the right side of the structure. The pipe cover and slope rule determines the optimal connection elevation for the new incoming pipe.

**Example 3: Existing Outgoing Pipes with New Incoming Pipe**
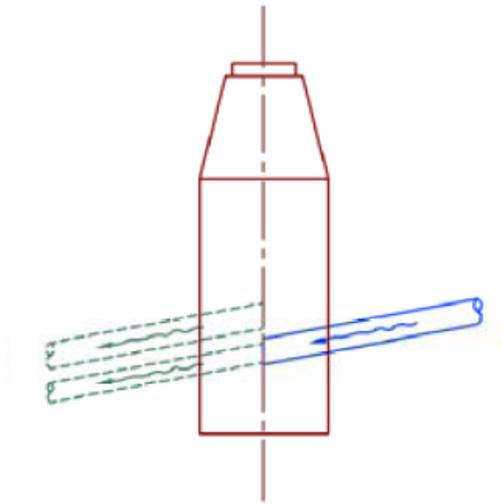


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The following illustration shows a junction structure that has two outgoing pipes already connected to it on the left side of the structure. A new outgoing pipe is added to the right side of the structure. The pipe cover and slope rule determines the optimal connection elevation for the new outgoing pipe.

**Example 4: Existing Outgoing Pipes with New Outgoing Pipe**
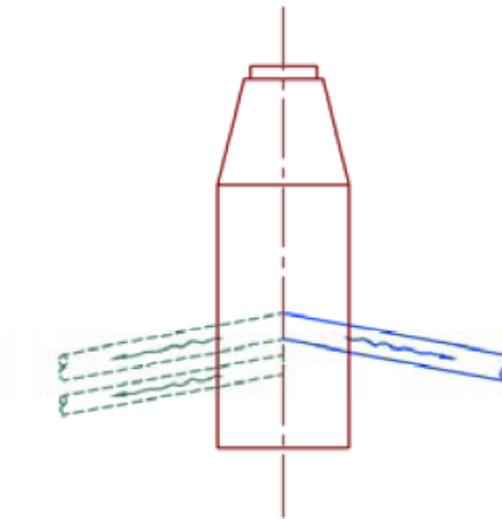


The following illustration shows a junction structure that has four pipes already connected to it on the left side of the structure. Two of those pipes are incoming pipes, and the other two are outgoing pipes. A new *incoming* pipe is added to the right side of the structure. The pipe cover and slope rule determines the optimal connection elevation for the new incoming pipe.

**Example 5: Existing Incoming and Outgoing Pipes with New Incoming Pipe**
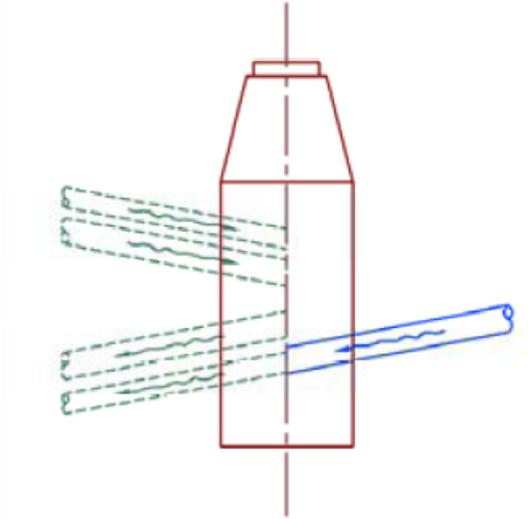
The following illustration shows a similar situation with both incoming and outgoing pipes already connected. In this case, a new *outgoing* pipe is added to the structure. The pipe cover and slope rule determines the optimal connection elevation for the new outgoing pipe.

**Example 6: Existing Incoming and Outgoing Pipes with New Outgoing Pipe**
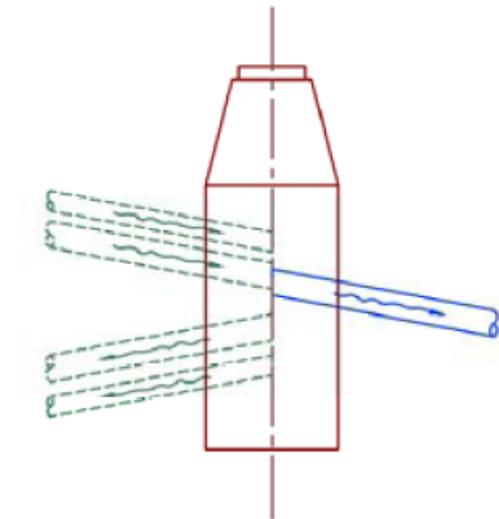
## Maximum Pipe Size Check Rule

This rule checks to see if pipes entering a structure have a diameter or width that exceeds a specified maximum value.

This rule analyzes all pipes attached to a structure and checks to see whether the pipe diameter or width is within the specified maximum value. For example, a manhole that is three feet wide in diameter may not be able to accommodate a pipe that is three feet wide in diameter.

### Parameters

The following parameters govern the behavior of this rule:

■ **Maximum Pipe Diameter or Width.** For circular pipes, this parameter measures the pipe diameter. For rectangular pipes, it measures width. If a pipe diameter or width exceeds the maximum value, a warning is issued on the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

## Set Sump Depth Rule

This rule sets the sump elevation of junction structures based on the user-specified parameters and the invert elevations of pipes attached to the structure.

When this rule is applied, the elevation of the inside bottom (sump) of the structure is calculated by subtracting the user-specified sump depth from the elevation of the lowest invert in the structure.

### Parameter

The following parameter governs the behavior of this rule:

■ **Sump Depth.** Specifies the sump depth, or the vertical distance from the invert of the lowest pipe attached to the structure to the inside bottom of the structure.

# Editing Part Rules

Use the Rules tab of either the Pipe Properties or the Structure Properties dialog box to edit the values for rules associated with a part in a pipe network.

Editing the rule values for a part in a drawing affects only the currently selected part in your *drawing*. It does not affect the definition of the part as it exists in the pipe network *part catalog*.

You can also edit rules as they are defined for a rule set in the Settings tree. For example, in the Toolspace Settings tree ➤ Pipe ➤ Pipe Rule Set collection, you can add rules, remove rules, or edit rule values for a particular rule set. This provides a convenient method for creating rule sets intended for a specific pipe network or project.

For example, you could create a rule set that has the Length Check rule value set to a certain value. You could do the same for the Cover and Slope rule.

**To edit rule values for a part in a pipe network**

1 In the drawing, right-click either a pipe or structure in a pipe network, and then click either Pipe Properties or Structure Properties.

2 Click the Rules tab.

3 Select the value of the rule you want to edit and enter a new value.

4 Click Apply or OK to save the changes.

**To add, remove, or edit rules in a rule set**

1 In Toolspace, on the Settings tab, expand either the Pipe or Structure collection.

**2** Expand either the Pipe Rule Set or Structure Rule Set collection

**3** Double-click a rule set. The Pipe or Structure Rule Set dialog box is displayed.

**4** Click the Rules tab.

Add or delete rules by clicking Add Rule or Delete Rule. Edit a rule value by clicking on the value and entering a new value.

**5** Click Apply or OK to save the changes.

## Overriding Part Rules

Override part rules if you do not want certain conditions flagged with warnings in the Prospector list view and in the Event Viewer.

You can select a part in a pipe network and override the rule value that governs it.

**To override rules from a rule set**

**1** In the drawing, right-click a pipe network part (either a pipe or structure), and then select either Pipe Properties or Structure Properties.

**2** Click the Rules tab.

**3** Clear the Use Values From Rule Set check box.

**4** Click Apply or OK to save the changes.

## Creating a Rule Set

Use rule sets to create customized rule behavior for pipe network parts.

A default "rule set" exists for both pipes and structures. The rule set simply provides a way for you to create a set of customized rules. You can add or exclude certain rules, or edit certain rule values, and save them in a named rule set designed to control the behavior for a specific pipe network or project.

For more information, see Part Rules (page 1247), the Pipe Rule Set dialog box (page 2037), or the Structure Rule Set dialog box (page 2050)

**To create a rule set**

**1** In Toolspace, on the Settings tab, expand either the Pipe or Structure collection.

**2** Right-click either Pipe Rule Set or Structure Rule Set. Click New.

**3** On the Rule Set Information tab, enter a name for the new rule set. Optionally, enter a description.

**4** Click the Rules tab.

**5** Click Add Rule to add a rule.

**6** In the Add Rule dialog box, in the Rule Name drop-down list, select a rule to add.

When you select a rule, its parameters are displayed in the Rule Parameters table.

**7** Click OK in the Add Rule dialog box.

**8** Edit a rule value by clicking a value on the Rules tab and entering a new value, or select one from a list.

**9** Click Apply or OK to save the changes.

## Converting VBA Rules to .NET

In drawings, you can convert AutoCAD Civil pipe and structure rules that are VBA-based to .NET.

In Civil 3D 2008, and in earlier versions of Civil 3D, the pipe and structure rules that were included in the product were created using VBA, and maintained in a .dvb file (*C3DPipeNetworkRules.dvb*).

In Civil 3D 2009, new pipe network rule sets that are created now point to the .NET dll version of rules, and therefore point to a .NET .dll file (*C3DPipeNetworkRules.dll*).

The .NET-based rules typically provide optimal performance over VBA-based rules. Civil 3D 2009, and later versions, still support VBA-based rules.

If you have not made any modification to your VBA pipe networks rules file (*C3DPipeNetworkRules.dvb*), it is recommended that you convert the pipe and structure rules in drawings to .NET using this command. If you have made edits to the C3DPipeNetworkRules.dvb rules file, it is not recommended that you convert rules to .NET because you will lose any VBA changes you made to *C3DPipeNetworkRules.dvb*.

If you create a new rule set in AutoCAD Civil 2009 or later, the rules will point to the .NET dll version of rules. If you open a drawing that has a rule set that was created in Civil 3D 2008, or earlier versions of Civil 3D, the rule set will still point to and execute the VBA version of rules, unless you run the convert to .NET command.

Please note the following:

■ This command only converts the VBA-based AutoCAD Civil pipe and structure rules that were included with the product. It does not convert any VBA custom rules you may have created behind the scenes.

■ By default, the rules files are installed to the following location:

■ Internally, this command just changes a pipe rule reference to the .NET version of that rule, instead of the VBA version of that rule.

■ In drawings, you can have a mix of pipe networks that point to VBA-based and .NET-based part rules. For example, when you open a drawing that was created in Civil 3D 2008 or earlier in Civil 3D 2009, that contains pipe networks, and then create a new rule set, that drawing will contain references to both VBA and .NET rules.

■ In a single pipe network, you can have a mix of parts that point to VBA-based and .NET-based part rules.

**To convert pipe and structure rules in a drawing from VBA to .NET**

1  Click Modify tab ➤ Design panel ➤ ▼ ➤ Convert VBA Pipe And Structure Rules To .NET.

    You are prompted to make sure that you really want to convert all pipe and structure rules in the drawing to .NET.

2  Click Yes.

    All VBA pipe and structure rules in the drawing are converted to VB.NET. Prompts indicate that the command has completed, and that the Event Viewer displays more details about the conversion.

### Quick Reference

Ribbon

Click Modify tab ➤ Design panel ➤ ▼ ➤ Convert VBA Pipe And Structure Rules To .NET

Menu

Click Pipes menu ➤ Utilities ➤ Convert VBA Pipe And Structure Rules To .NET.

Command Line

ConvertVBARulesToDot.Net

# Labeling Pipe Networks

You can add labels to pipe network parts either when you create the objects or after you create them.

Use the Create Pipe Network dialog box (page 2023) to specify label styles for pipes and structures in plan view. Use the Pipe Network Properties dialog box (page 2033) to change plan view label styles, or to specify label styles for profile and section views. If you select a label style of <none>, labels are not added.

Use the Add Pipe Network Labels option to access the Add Labels Dialog Box (page 1970) if you need to create or modify a style before labeling a pipe network.

Pipe network label types you can add after creation include entire network, single part, and spanning labels.

If desired, you can add a Material type to a pipe or structure label, as long as the Material property has been added to the part family that the pipe or structure part belongs to. See Adding a Material Property to a Part (page 1322) for more information. You can also label hydraulic properties in pipe networks. See Labeling Hydraulic Properties in Pipe Networks (page 1270) for more information.

You can use the following commands to add labels to pipe network parts.

| Command | Description |
| --- | --- |
| AddNetworkPartPlanLabel | Adds labels to pipes and structures in plan view. |
| AddNetworkPartProfLabel | Adds labels to pipes and structures in profile view. |
| AddNetworkPartSectLabel | Adds labels to pipes and structures in section view. |
| AddNetworkPlanLabels | Adds labels to a pipe network in plan view. |
| AddNetworkProfLabels | Adds labels to a pipe network in profile view. |
| AddNetworkSectLabels | Adds labels to a pipe network in section view. |
| AddSpanningPipeLabel | Adds label to a span of connected pipes. |

For more information, see Understanding Labels in AutoCAD Civil (page 1466).

# Labeling Multiple Pipes and Structures in Plan Views

You can label pipe and structures in plan views with Pipe label styles.

**To label all pipes and structures within an entire network in plan views**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Plan .

2 Select the part contained in the network to be labeled. All pipes or structures will be labeled in the network.

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Entire Network Plan.

**4** Specify the other options as required.

**5** Click Add and then select the part to label.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Plan 🏷

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

Command Line

AddNetworkPlanLabels

## Labeling Multiple Pipes and Structures in Profile Views

You can label pipe and structures in profile views with Pipe label styles.

**To label pipes and structures in profile views**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Profile 🏷 .

**2** Select the part contained in the network to be labeled. All pipes or structures in the network will be labeled.

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels 🏷 .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Entire Network Profile.

**4** Specify the other options as required.

**5** Click Add and then select the part to label.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Profile 🏷

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

Command Line

AddNetworkProfLabels

# Labeling Multiple Pipes and Structures in Section Views

You can label pipes and structures in section views with Pipe label styles.

**To label pipes and structures in section views**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Section ✎ .

2 Select the part contained in the network to be labeled. All pipes or structures in the network will be labeled.

OR

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ✎ .

2 In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

3 Under Label Type, select Entire Network Section.

4 Specify the other options as required.

5 Click Add and then select the part to label.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Entire Network Section ✎

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

Command Line

AddNetworkSectLabels

# Labeling Individual Pipes and Structures in Plan Views

You can label individual parts in plan views with Pipe label styles.

**To label pipe networks in plan views**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Plan ✎ .

2 Select the part contained in the network to be labeled. The individual part will be labeled.

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ✎ .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Single Part Plan.

**4** Specify the other options as required.

**5** Click Add and then select the part to label.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Plan ✎

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

Command Line

AddNetworkPartPlanLabel

## Labeling Individual Pipes and Structures in Profile Views

You can label individual parts in profile view with Pipe label styles.

**To label pipe networks in profile view**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Profile ✎ .

**2** Select the part contained in the network to be labeled. The individual part will be labeled.

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ✎ .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Single Part Profile.

**4** Specify the other options as required.

**5** Click Add and then select the part to label.

### Quick Reference

Ribon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Profile ✎

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

AddNetworkPartProfLabel

# Labeling Individual Pipes and Structures in Section Views

You can label individual parts in section view with Pipe label styles.

**To label pipes and structures in section view**

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Section ✎ .

2   Select the part contained in the network to be labeled. The individual part will be labeled.

OR

1   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ✎ .

2   In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

3   Under Label Type, select Single Part Section.

4   Specify the other options as required.

5   Click Add and then select the part to label.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Single Part Section ✎

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Add Pipe Network Labels

Command Line

AddNetworkPartSectLabel

# Labeling a Span of Connected Pipes

In plan or profile views, you can label a series of consecutively connected pipes with a single pipe label.

A spanning label helps minimize the number of labels displayed in a pipe network.

When you place the cursor over the spanning pipe label, the pipes that are in the span are highlighted. You can also use the AutoCAD list command to list the connected pipes. After adding a spanning label, you can highlight the connected pipes that comprise the span by selecting the label.

**See also:**

■   Resetting the Anchor Pipe on Spanning Labels (page 1270)

**To label a span of pipes in a plan view**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Spanning Pipes Plan ✎ .

**2** At the command line, you are prompted to select the first pipe network part (pipe or structure).

**3** In the drawing, click a pipe or structure in the pipe network to define the starting point for the connected pipe span.

**4** At the command line, you are prompted to select the next pipe network part (pipe or structure).

**5** Click a part (pipe or structure) to define the ending point for the span.

**6** Press Enter.

The following prompt is displayed:

Specify label location on pipe:

**7** Specify the location for the label by clicking one of the pipes in the span. You must click a pipe, not a structure.

If you select a pipe that is not included in the span, a prompt is displayed indicating that the label location must be on a spanned pipe.

Note that the label can be slid along the span with the slider grip ◥ .

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ✎ .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Spanning Pipes Plan.

**4** Specify the other options as required.

**5** Click Add.

You are prompted to select the first part (pipe or structure) in the span.

**6** Define the span by clicking the first and last parts (pipes or structures) in the span. The parts must belong to the same pipe network.

**7** Press Enter.

The following prompt is displayed:

Specify label location on pipe:

**8** Specify the location for the label by clicking one of the pipes in the span. You must click a pipe, not a structure.

If you select a pipe that is not included in the span, a prompt is displayed indicating that the label location must be on a spanned pipe.

Note that the label can be slid along the span with the slider grip ◥ .

**To label a span of pipes in a profile view**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Spanning Pipes Profile ✎ .

**2** At the command line, you are prompted to select the first pipe network part (pipe or structure).

**3** In the drawing, click a pipe or structure in the pipe network to define the starting point for the connected pipe span.

**4** At the command line, you are prompted to select the next pipe network part (pipe or structure).

**5** Click a part (pipe or structure) to define the ending point for the span.

**6** Press Enter.

The following prompt is displayed:

Specify label location on pipe:

**7** Specify the location for the label by clicking one of the pipes in the span. You must click a pipe, not a structure.

If you select a pipe that is not included in the span, a prompt is displayed indicating that the label location must be on a spanned pipe.

Note that the label can be slid along the span with the slider grip ◣ .

OR

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Add Pipe Network Labels ◈ .

**2** In the Add Labels dialog box (page 1970), under Feature, ensure that Pipe Network is selected.

**3** Under Label Type, select Spanning Pipes Profile.

**4** Specify the other options as required.

**5** Click Add.

You are prompted to select the first part (pipe or structure) in the span.

**6** Define the span by clicking the first and last parts (pipes or structures) in the span. The parts must belong to the same pipe network.

**7** Press Enter.

The following prompt is displayed:

Specify label location on pipe:

**8** Specify the location for the label by clicking one of the pipes in the span. You must click a pipe, not a structure.

If you select a pipe that is not included in the span, a prompt is displayed indicating that the label location must be on a spanned pipe.

Note that the label can be slid along the span with the slider grip ◣ .

**To highlight a connected pipe span, do one of the following**

■ Hover the cursor over the spanning label. This highlights all of the parts that are in the span.

■ Right-click the pipe span label and select Show Spanning Pipes. A dotted line will appear from the starting point to the end point of the connected pipe span, as shown in the following illustration. The diamond-shaped slider grip can be used to slide the label along the span. To remove the highlight, deselect Show Spanning Pipes from the right-click menu.
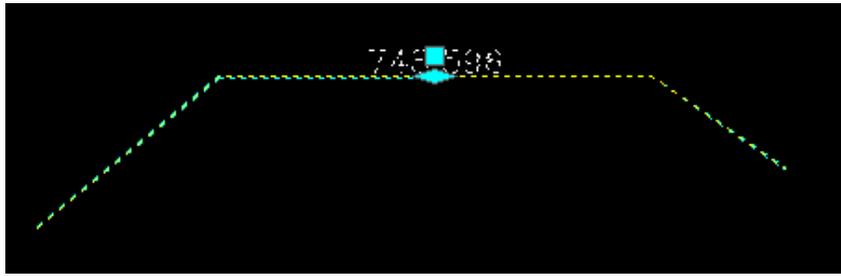
### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Pipe Network ➤ Spanning Pipes Plan 

Menu

Pipes menu ➤ Add Pipe Network Labels ➤ Spanning Pipes Plan

Command Line

AddNetworkLabels

## Editing Spanning Labels with Grips

Spanning labels are not anchored to any particular pipe or structure. You can move them freely to any connected pipe or structure in the span.

It is also possible to reset the anchor point of a spanning label. For more information, see Resetting the Anchor Pipe on Spanning Labels (page 1270).

**To move a spanning pipe label**

■ Click the label to be moved and select the slider grip  .

Move the label to the desired location.

---

**NOTE** Spanning pipe labels have the same dragged state behavior as regular pipe labels. Click the round grip  to restore a dragged label to its default state.

---

## Deleting Pipes Labeled with Spanning Labels

Deleting an individual pipe from a span labeled with a spanning label has various affects depending on where the label is located in relation to the deleted pipe.

To delete a pipe, select it and press delete.

In the following example, the pipe span is comprised of four connected pipes. Note where the spanning label is located. In this example, the Text Component Editor was used to create a label that dynamically indicates the number of pipes in the span by inserting the Number of Pipes in Span property. For more information see, Text Component Editor Dialog Box (page 1961).
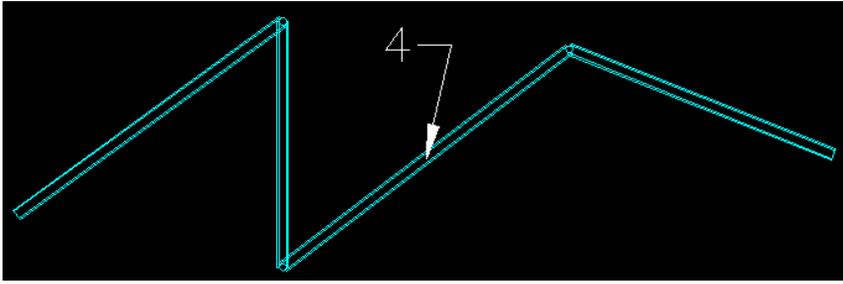
**Example 1**

In the following example, the vertical pipe has been deleted, or detached from the other pipes. The result is that only the connected pipes to the right are now included in the span label. The label stays at the same location.
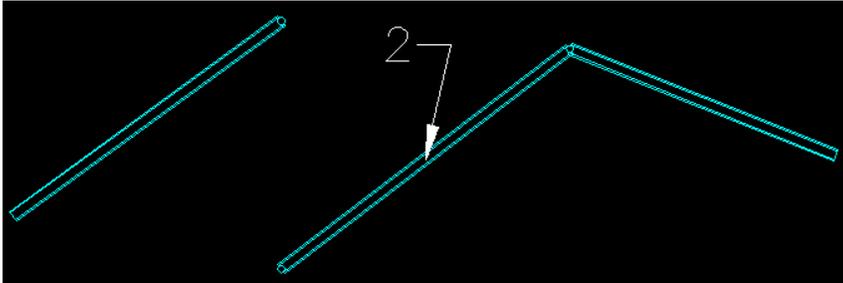


**Example 2**

In the following example, the pipe to the left has been deleted, or detached from the vertical pipe. The result is that only the remaining pipes are included in the span. The label indicates the span is now comprised of three pipes.
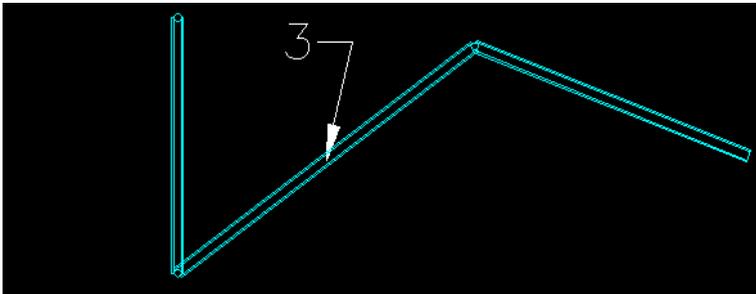


**Example 3**

In the following illustration, the pipe with the span label has been deleted. The result is that the span label is also deleted.

## Resetting the Anchor Pipe on Spanning Labels

You can reset the anchor position of label that spans multiple parts in a pipe network.

**To reset the anchor position of a label that spans multiple pipe network parts**

1   At the command line, enter ResetAnchorPipe.

2   In the drawing, select a label that spans multiple pipe parts (spanning pipe label).

3   Select the new anchor point by clicking on a pipe in the same network. You must select a pipe, not a structure.

    The new anchor position is set, and the spanning label will display at the midpoint of the selected pipe.

## Adding a Material Type to a Pipe or Structure Label

You can add a material type, such as ductile iron or corrugated steel, to display in a pipe or structure label.

If the part family the pipe or structure belongs to includes the Material property, you can configure that property to display text identifying the type of material used for that part. Using the Label Style Composer, you can add that Material property to labels for pipe network parts.

**To add a material type to an existing pipe or structure label**

1   In the drawing, select a pipe or structure and then click Pipe Properties or Structure Properties.

2   On the Pipe Properties dialog box, or the Structure Properties dialog box, click the Part Properties tab.

3   Scroll down to the Part Data section and the Material property should be displayed.

    If the Material property is not displayed, this means that this part family does not contain the Material property. See Adding a Material Property to a Part (page 1322) for instructions on how to add this property to a part family.

4   Click in the Value cell to select an existing or enter a new value for the Material property.

5   Click OK on the Pipe Properties dialog box or the Structure Properties dialog box.

**To add a material type to a pipe or structure label style**

1   In the Label Style Composer, click the Layout tab.

2   In the Text section, click in the Contents cell, and then ⊡ click to display the Text Component Editor.

3   In the Text Component Editor, select Material in the Properties list and add it to your label.

4   Click OK on the Text Component Editor dialog box.

5   Click OK on the Label Style Composer dialog box.

## Labeling Hydraulic Properties in Pipe Networks

You can create a label style for labeling hydraulic properties, such as HGL and EGL, in pipe networks.

Use the Label Style Composer to create a label style that lets you add labels that display values for hydraulic properties, such as hydraulic grade line and energy grade line in pipe networks.

Once the label style is created, you can select that label style on the following tabs of the Pipe Network Properties dialog box: Layout Settings tab, Profile tab, Section tab.

**To create a label style for pipe network hydraulic properties**

**1** In Toolspace, on the Settings tab, expand the Pipe collection, then expand the Label Styles collection.

**2** In the Label Styles collection, expand a collection. For example, expand the Plan Profile collection , and then right-click on an existing label style. For example, right-click on Standard, and then click New or Edit.

**3** On the Label Style Composer, click the Layout tab.

**4** Under the Text collection, click the Browse button [icon] in the Value cell next to the Add Contents property.

**5** Use the Text Component Editor to add the desired hydraulic properties to the label style. For example, on this dialog box, under Properties, click the drop-down arrow to display the list of properties you can add to the label. Select the desired hydraulic properties from this list. For example, click Hydraulic Grade Line Up, and then click the right-arrow button to add the property to the label style.

**6** When finished composing label text using this dialog box, click OK to save changes.

**7** Click OK on the Label Style Composer dialog box to save changes to the label style.

For more information, see Text Component Editor Dialog Box (page 1961)or Adding Content to Labels (page 1502).

### Quick Reference

Dialog Box

Text Component Editor (page 1961)

# Adding Pipe Tables

You can create tables that display information about pipes inone or more a pipe networks.

When you insert a pipe table into a drawing, specified pipe information is automatically displayed in the table.

Pipe tables can include data such as pipe shape, size, length, slope, start and end stations, or any pipe property.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. The procedure in this topic explains how to access the pipe table creation command. It provides a link to information about the Pipe Table Creation dialog box. For information about modifying tables, see Modifying Tables (page 1567).

**To create a pipe table**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Pipe [icon] .

**2** In the Pipe Table Creation (page 2075) dialog box, change the generic table settings as needed.

**3** Select a network to be included in the table.

**4** Click OK.

**5** Select the location for the upper-left corner of the table in the drawing.

### Quick Reference

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Pipe ⊞

Menu

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Pipe ⊞

# Adding Structure Tables

You can create tables that display information about structures in one or more pipe networks.

When you insert a structure table into a drawing, specified structure information is automatically displayed in the table.

Structure tables can include data such as structure type, elevation, sump depth, station, or any structure property.

You create and edit tables for most objects using the same common procedures and standard dialog boxes. The procedure in this topic explains how to access the structure table creation command. It provides a link to information about the Structure Table Creation dialog box. For information about modifying tables, see Modifying Tables (page 1567).

**To create a structure table**

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Structure ⊞ .

**2** In the Structure Table Creation (page 2076) dialog box, change the generic table settings as needed.

**3** Select a structure to be included in the table.

**4** Click OK.

**5** Select the location for the upper-left corner of the table in the drawing.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Structure ⊞

Menu

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Pipe Network ➤ Add Structure ⊞

# Editing Structure Table Styles

Edit structure table styles in the Table Styles collection in Toolspace, on the Settings tab.

Structure table styles govern how structure tables are displayed in a drawing. You can create, edit, or delete structure table styles. Create an entirely new table style, or create a table style based on an existing style.

You can compose a table cell with more than one text component in order to show connected pipe data for a given structure. The text components display data for each object associated with a parent object (the

object you are storing data for). For example, you can use a component in a structure table style to include cell components for pipes associated with that structure.

**To edit a structure table style**

1  In Toolspace, on the Settings tab, expand Structure ➤ Table Styles.

2  Right-click the table style that you want to edit and click Edit.

3  In the Table Style dialog box, modify style settings.

**To add table cell components**

1  Double-click a Structure Table cell in the Data Properties tab of the Table Style dialog box to access the Table Cell Components dialog box.

2  Click the down arrow in $A_+$ , and then click $\oplus$ to create a new text component.

   You can add any number of components, and there can be more than one of any given component type. The possible components are: Structure, Structure All Pipes, Structure In Flow Pipes, and Structure Out Flow Pipes.

3  Click Text $\boxed{\cdots}$ to access the Text Component Editor Dialog Box (page 1961). When this option is selected, the component is added to the list of available components. You can edit the name of the component and/or enter the content for the table cell.

4  Set the cell justification.

5  Click OK.

## Quick Reference

Toolspace Shortcut Menu

   Settings: Right-click Structure table style ➤ New

Dialog Box

   Table Style (page 2437)

# Rendering Pipe Network Parts

Render pipe network parts using the AutoCAD Render command.

The render material that is assigned to a pipe network part by default, upon creation, is defined in the parts list. After the part is added to the drawing, you can edit the render material for the part using the Render Material option on the Information tab of either the Pipe Properties or the Structure Properties dialog box.

For more information about rendering AutoCAD Civil objects and data, see Rendering Objects (page 1579).

**To render pipe network parts**

1  Create a pipe network.

2  Do one of the following:
   ■  In Toolspace, on the Prospector tree, expand the Pipe Networks collection, then expand the desired pipe network, and click either Pipes or Structures. In the Prospector list view, choose a render material for the desired parts.

- In the drawing, right-click the pipe network part and click Properties. On either the Pipe Properties or the Structure Properties Information tab, select a render material.

- Use the AutoCAD Render command to render the pipe network. For more information, see Render a Model in the AutoCAD Help.

# Using Hydraulics and Hydrology Features

The Hydraflow extensions allow you to peform a variety of hydraulics and hydrology analysis tasks on pipe networks created with AutoCAD Civil or with the Hydraflow Storm Sewers Extension.

You can migrate pipe network data between AutoCAD Civil and the Hydraflow Storm Sewers Extension.

**NOTE** The Hydraflow extensions (Storm Sewers, Hydrographs, and Express) are included with AutoCAD Civil 3D; however, they are not included with AutoCAD Civil.

## Moving Pipe Network Data Between AutoCAD Civil and Storm Sewers

You can migrate pipe network data between AutoCAD Civil and the Storm Sewers Extension.

For example, with AutoCAD Civil, you can:

- export one or more AutoCAD Civil pipe networks to an *.stm* file that can then be opened by the Hydraflow Storm Sewers Extension for analysis and or editing

- import one or more pipe runs from an *.stm* file that has been exported from the Hydraflow Storm Sewers Extension into AutoCAD Civil

When importing Storm Sewer data into AutoCAD Civil, you can import a single pipe network, or multiple pipe networks. However, you must import entire pipe networks; you cannot import individual parts, or a series of connected parts within a pipe network.

Before migrating pipe network data between AutoCAD Civil and the Storm Sewers Extension, there are a few preparation tasks you must perform. For more information, see Preparing for Migrating Storm Sewers Data (page 1275).

**Importing STM Files From Previous Versions**

In AutoCAD Civil, you can import *.stm* files that were created or saved from older versions of the Storm Sewers Extension; for example, the Storm Sewers Extension for AutoCAD Civil 3D 2009 or earlier. You can also successfully open *.stm* files that were created or saved from 2009, or earlier, versions of the Storm Sewers Extension in the Storm Sewers Extension version 2010. However, in the Storm Sewers Extension for Civil 3D 2009, or earlier versions of the Storm Sewers Extension, you cannot open *.stm* files that were saved in the Storm Sewers Extension for Civil 3D 2010, or that were exported from AutoCAD Civil.

It is also important to note that when importing *.stm* files that were saved prior to version 2010, they may contain hydraulic grade line (HGL) values, however, they will not contain some of the other hydraulic properties, as listed below. The following values are calculated in the Storm Sewers Extension version 2010, but are not found in older versions of *.stm* files (prior to 2010):

- Pipes

  - Energy Grade Line Up

  - Energy Grade Line Down

- ■ Flowrate

- ■ Structures

  - ■ Energy Grade Line
  - ■ Known Capacity

**Storm Sewers Pipe Size Calculations**

In the Storm Sewers Extension, the list of pipe sizes is hard coded and cannot be customized. This list does not distinguish between inner and outer pipe diameter, pipe material, or pipe class, as does AutoCAD Civil. When you perform calculations that resize pipes in the Storm Sewers Extension, the results include only the sizes that are available in the hard coded list. This may result in a pipe size or sizes that are not typical of the pipe material they are working with. However, you can always manually edit the pipe size. For more information, see the Storm Sewers Extension Help.

# Preparing for Migrating Storm Sewers Data

There are a few preparatory tasks to perform before migrating data between AutoCAD Civil and the Storm Sewers Extension.

The table below summarizes these tasks.

| Task | Description |
|------|-------------|
| Verify that your pipe network data meets the necessary criteria for migration | Review the criteria for migrating pipe network data. For more information, see Criteria for Migrating Pipe Networks (page 1276). |
| Create the parts list that will be used during migration | When you import or export pipe networks between AutoCAD Civil and the Storm Sewers Extension, a parts list that contains appropriate parts for the import or export is required. You will need to create this parts list first, before you begin importing or exporting the data. You may need to create a full parts list, containing all of the available parts in the entire catalog, or you may need to create a smaller, customized parts list that contains only the parts included in the migrated data. For more information, see Creating a Full Parts List (page 1239) or Creating a Parts List (page 1238). |
| Set the part matchup settings that will be used during migration | In Toolspace, on the Settings tab, you need to configure defaults that will be used for matching up part types between AutoCAD Civil and the Storm Sewers. For more information, see Setting Part Matching Defaults for Migration (page 1276) . |
| Choose the default parts list that will be used during migration | In Prospector, on the Settings tab, you also need to choose the default parts list that will be when you migrate data between AutoCAD Civil and the Storm Sewers. For more information, see Setting the Default Parts List For Migration (page 1277). |

Optionally, you may want to create a label style for displaying labels for hydraulic properties, such as hydraulic grade lines and energy grade lines in pipe networks. For more information, see Labeling Hydraulic Properties in Pipe Networks (page 1270).

## Criteria for Migrating Pipe Networks

To successfully migrate pipe network data between the Storm Sewers Extension and the AutoCAD Civil, the pipe network data must meet certain conditions.

Verify that your pipe network data meets these criteria before migrating:

■ Note that in the Storm Sewers Extension, a pipe and its corresponding upstream inlet structure make up a Hydraflow Storm Sewers line.

■ The Storm Sewers Extension can accept a maximum of 250 pipes (lines) with structures in one project. If a single AutoCAD Civil pipe network contains more than 250 pipes (lines), then the pipe network must be divided into smaller pipe networks (and therefore, multiple .stm files) before the pipe network data can be opened in the Hydraflow Storm Sewers Extension.

■ All catch basin and junction structures must have an outlet pipe. To verify this, in AutoCAD Civil display the Connected Pipes tab of the Structure Properties dialog box. Look at the In/Out column.

It is also important to be aware of the following before exporting pipe networks to an .stm file (or any type of export of pipe data):

■ If a pipe does not have an upstream structure in AutoCAD Civil, it is given a null structure in the exported file.

■ If a structure is not attached to a pipe, the structure will not be exported.

## Setting Part Matching Defaults for Migration

You must set part matching defaults before migrating pipe network data between AutoCAD Civil and Storm Sewers.

Setting the part matching defaults enables you to choose the specific AutoCAD Civil part types that will be matched up with Storm Sewers part types when you import and export pipe network data between AutoCAD Civil and Storm Sewers.

Performing this task helps avoid error messages during migration, and ensures that AutoCAD Civil part types are recoginized as appropriate part types in the Storm Sewers Extension. and vice versa.

You may only have to perform this procedure once. However, if you are adding, deleting, or changing part types after import or export, then you would need to perform this procedure again to ensure that parts are appropriately matched between the two applications.

**To set part matching defaults for migrating pipe network data**

1 In Toolspace, on the Settings tab, right-click the Pipe Network collection and select Edit Feature Settings.

2 In the Edit Feature Settings - Pipe Network dialog box, expand Storm Sewers Migration Defaults.

3 For the Part Matching Defaults, click the Browse button in the V ⬚ alue cell to display the Part Matchup Settings - Storm Sewers Settings dialog box (page 2079).

4 To set the defaults that will be used when you are **importing** data from the Storm Sewers Extension into AutoCAD Civil, click the Import tab.

5   On the Import tab, expand each part type category in the first column, and click on an item in the column on the right to display the Browse button $\boxed{\cdots}$. For example, click in the Concentric Cylindrical Structure cell.

6   Click $\boxed{\cdots}$ to display the Part Catalog dialog box.

7   On the Part Catalog dialog box, select the desired AutoCAD Civil part type and then click OK.

8   On the Import tab, scroll down to ensure that all part types in the first column are matched up with appropriately part types in the second column.

9   To set the part type defaults that will be used when you are **exporting** data from AutoCAD Civil into the Storm Sewers Extension, click the Export tab.

10  On the Export tab, click in the column on the right to select appropriate parts from the drop-down list.

11  On the Export tab, scroll down to ensure that all part types in the first column are matched up with appropriately part types in the second column.

12  When you are finished matching part type defaults, click OK.

Once you have finished matching up part types, the next step is to select the parts list that will be used for migration by default. See Setting the Default Parts List For Migration (page 1277) for more information.

## Setting the Default Parts List For Migration

You must set the default parts list that will be used during migration before migrating pipe network data between AutoCAD Civil and Storm Sewers.

Typically, you will want to create a parts list that contains all of the needed prts for the import or export. You may need to create a full parts list, containing all of the available parts in the entire catalog, or you may need to create a customized, smaller, more manageable parts list that contains only the parts included in the migrated data. For more information, see Creating a Full Parts List (page 1239) or Creating a Parts List (page 1238)

**To set the default parts list for migration**

1   In Toolspace, on the Settings tab, right-click the Pipe Network collection and select Edit Feature Settings.

2   In the Edit Feature Settings - Pipe Network dialog box, expand Storm Sewers Migration Defaults.

3   For Parts List Used For Migration, click the Browse button in the Value cell to select a parts list.

4   Click OK to select the parts list.

When pipe network data is imported into or exported from AutoCAD Civil, this is the parts list that will be used to search for part families and part sizes.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Pipe Network collection ➤ Edit Feature Settings ➤ Storm Sewers Migration Defaults

Dialog Box

Part Matchup Settings Dialog Box (page 2079)

# Importing Pipe Network Data from an .stm File

You can import pipe network data from a Storm Sewers Extension *.stm* file into AutoCAD Civil.

Pipe networks imported from the Storm Sewers Extension can contain hydraulic property information (page 1280), such as hydraulic grade lines (HGL) and energy grade lines (EGL) and more.

When you import the .stm file, if any of the pipe network names in the *.stm* file match pipe network names in the AutoCAD Civil drawing, you must choose to either create new pipe networks, or update existing pipe networks.

If there are no name conflicts, the new pipe networks are added to the AutoCAD Civil drawing with their existing names.

Before importing an *.stm* file, it is important to understand issues associated with importing *.stm* files that were saved prior to version 2010. For more information, see Moving Pipe Network Data Between AutoCAD Civil and Storm Sewers (page 1274).

**NOTE** It is important to note that rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Storm Sewers Extension. For more information about rules, see Part Rules (page 1247).

**Matching Pipe Network Names**

In cases where the *.stm* file contains one or more pipe network names that match pipe network names in the currently open AutoCAD Civil drawing, you will be prompted to choose one of the following options:

**Create new pipe network:** When you choose this option, the pipe network will be imported into the drawing as a new pipe network. The name of the newly imported pipe network is appended with a numeric counter (1). For example, StormSewerNetwork1(1).

**Update the existing pipe network:** When you choose this option, the pipe network will be imported into the drawing and it will update the existing pipe network that has the same pipe network name. For example, if you have a pipe network named StormSewerNetwork1 in the drawing, and you import an *.stm* file that contains a pipe network named StormSewerNetwork1, then the StormSewerNetwork1 imported from the *.stm* file updates (overwrites) the pipe network named StormSewerNetwork1 in the drawing.

**No Matching Pipe Network Names**

As previously mentioned, if the pipe network names contained in the *.stm* file do not match any pipe network names that exist in the currently open drawing, then the new pipe networks will be added to the drawing with their existing names. These new pipe networks will be displayed in the drawing and in the Prospector tree.

**To import an *.stm* file into AutoCAD Civil**

1 From the Home tab, click the Insert tab, and then click Import Storm Sewers .

2 On the Import Storm Sewers File dialog box, navigate to the desired *.stm* file, select it, and then click Open.

   You are prompted to either update the existing pipe network, or to create a new pipe network.

3 Choose to either update the existing pipe network, or to create a new pipe network.

**NOTE** It is important to note that rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Storm Sewers Extension. For more information about rules, see Part Rules (page 1247).

**Quick Reference**

Ribbon

Click Insert tab ➤ Import panel ➤ Storm Sewers 

Or

Click Pipe Networks contextual tab ➤ Analyze panel ➤ Storm Sewers drop-down ➤ Import File 

Menu

File menu ➤ Import ➤ Import Data From Storm Sewers

Command Line

ImportStormSewers

# Exporting Pipe Network Data to an .stm File

You can export pipe network data from AutoCAD Civil to an *.stm* file.

The *.stm* file can then be opened bythe Hydraflow Storm Sewers exstension for analysis and editing. After editing the file in Storm Sewers, you can import it back into AutoCAD Civil. Changes made to the file while in the Storm Sewers extention are recognized in AutoCAD Civil.

You can export one or more pipe networks into an *.stm* file. You must choose entire pipe networks. You cannot export individual parts, or a set of connected parts within a pipe network.

It is important to note the following before exporting pipe networks to an *.stm* file (or any type of export of pipe data):

- If a pipe does not have an upstream structure in AutoCAD Civil, it is given a null structure in the exported file.

- If a structure is not attached to a pipe, the structure will not be exported.

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**NOTE** You cannot open *.stm* files that were exported from AutoCAD Civil, or that were saved in the Storm Sewers Extension for Civil 3D 2010, in the Storm Sewers Extension for Civil 3D 2009, or earlier versions of the Storm Sewers Extension.

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**To export an *.stm* file from AutoCAD Civil**

1 Do one of the following:
   - In the drawing, select a pipe network. On the Pipe Networks tab, click the Analyze tab ➤ Storm Sewers drop-down ➤ Export To File.

   - Click the Output tab ➤ Export panel ➤ Export To File.

   - In Toolspace, on the Prospector tab, right-click the Pipe Networks collection, or right-click the pipe network you want to export, and then select Export to Storm Sewers  .

   - At the command line, enter ExportStormSewerData.

2 On the Export to Storm Sewers dialog box, specify the pipe networks to be exported, and then click OK.

3 On the Export Storm Sewers to File dialog box, specify the location and the name for the *.stm* file, and then click Save.

**To export an *.stm* file from AutoCAD Civil and open it in Storm Sewers for editing**

1   Do one of the following:

■ In the drawing, select a pipe network. From the Pipe Networks tab ➤ Analyze panel ➤ Storm Sewers drop-down ➤ Edit In Storm Sewers ⊘ .

■ At the command line, enter EditInStormSewers.

2   On the Export to Storm Sewers dialog box, specify the pipe networks to be exported, and then click OK.

3   On the Export Storm Sewers to File dialog box, specify the location and the name for the *.stm* file, and then click Save.

## Quick Reference

Ribbon

Click Output tab ➤ Export panel ➤ Export To Storm Sewers 🖳 .
OR
Click Pipe Networks contextual tab ➤ Analyze panel ➤ Storm Sewers drop-down ➤ Export to File ⊘

Menu

File menu ➤ Export ➤ Export Data From Storm Sewers

Toolspace Shortcut Menu

Prospector tab: right-click pipe network ➤ Export to Storm Sewers

Command Line

ExportStormSewers

Dialog Box

Export to Storm Sewers Dialog Box (page 2078)

## Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks

You can add hydraulic property data to AutoCAD Civil pipe networks in a few different ways.

Pipe networks created in AutoCAD Civil do not contain values for hydraulic property information, such as hydraulic grade lines (HGL) and energy grade lines (EGL), by default. While the pipe networks contain the properties for this type of information, the hydraulic properties are set to zero or are "not set" by default.

You can add hydraulic property data to a AutoCAD Civil pipe network using the following methods:

■ by exporting the pipe network to an *.stm* file, importing that file into the Storm Sewers Extension, and using the Storm Sewers Extension to add hydraulic properties to the pipe network
You can use the Storm Sewers Extension to perform the hydraulic analysis on the pipe run. Once you have done this, you can then import the *.stm* file back into AutoCAD Civil. At this point, the AutoCAD Civil pipe network will containing the valid hydraulic property data (HGL, EGL, and so on) that it got from the Storm Sewers Extension.

■ You can also obtain hydraulic property data by importing an .xml file that contains valid hydraulic property information.

■ You can also enter your own values for hydraulic properties using the Panorama. This may be useful if you intend to later import the pipe network data into some other application, such as the Storm Sewers Extension, to perform hydraulic analysis.

Once you have the hydraulic property data in a AutoCAD Civil pipe network, you can display, edit, and label the hydraulic grade lines and energy grade lines in profile views. You can also view and edit the specific hydraulic property data values on the pipe and structure Properties dialog boxes and vistas.

When you add hydraulic analysis data to AutoCAD Civil pipe networks, the following hydraulic properties are populated with valid values for pipes and structures:

| Hydraulic Property | Available for... |
| --- | --- |
| hydraulic grade line (page 2475) | pipes and structures |
| energy grade line (page 2473) | pipes and structures |
| flow rate (page 2474) | pipes |
| junction loss (page 2476) | pipes |
| return period (page 2481) | pipes |
| known capacity (page 2476) | structures |
| bypass target (page 2470) | structures |

In AutoCAD Civil, hydraulic grade lines and energy grade lines are displayed as a straight line through structures, connected between the HGL Up or EGL Up property from one pipe to the HGL Down or EGL Down property of the next pipe. See Viewing Hydraulic Data in Profile Views (page 1282) for an illustration of a hydraulic grade line and an energy grade line displayed in a profile view.

You can also create label styles that display labels for hydraulic properties, such as HGL and EGL. For more information, see Labeling Hydraulic Properties in Pipe Networks (page 1270).

**To add hydraulic property data to an AutoCAD Civil pipe network**

1 In AutoCAD Civil, export the pipe network to an *.stm* file. For more information, see Exporting Pipe Network Data to an .stm File (page 1279).

2 In the Storm Sewers extension, perform the hydraulic analysis tasks on the pipe network. This adds valid hydraulic property data (HGL, EGL) to the pipe network. For more information, see the Storm Sewers extension Help.

> **NOTE** The Hydraflow extensions (Storm Sewers, Hydrographs, and Express) are included with AutoCAD Civil 3D; however, they are not included with AutoCAD Civil.

3 Export the pipe network from Storm Sewers to an *.stm* file. For more information, see the Storm Sewers extension Help.

In AutoCAD Civil, import the *.stm* file that now contains valid hydraulic property data. For more information, see Importing Pipe Network Data from an .stm File (page 1278)

Verify that the AutoCAD Civil pipe network contains valid HGL and EGL data. See verifying hydraulic property data (page 1282).

**To verify hydraulic property data**

1 In AutoCAD Civil, right-click a part in the pipe network and select Pipe Properties or Structure Properties.

2 On the Pipe Properties or Structure Properties dialog box, click the Part Properties tab.

Note the items listed in the Hydraulic Properties section. The values listed here are set when the pipe network is imported from the *.stm* file. For more information on these properties, see the Pipe Properties dialog box (page 2039) or the Structure Properties dialog box (page 2051).

## Viewing Hydraulic Data in Profile Views

You can view hydraulic property data (hydraulic grade lines and energy grade lines) from pipe networks in profile views.

For example, the following illustration shows a hydraulic grade line and an energy grade line displayed in a profile view.
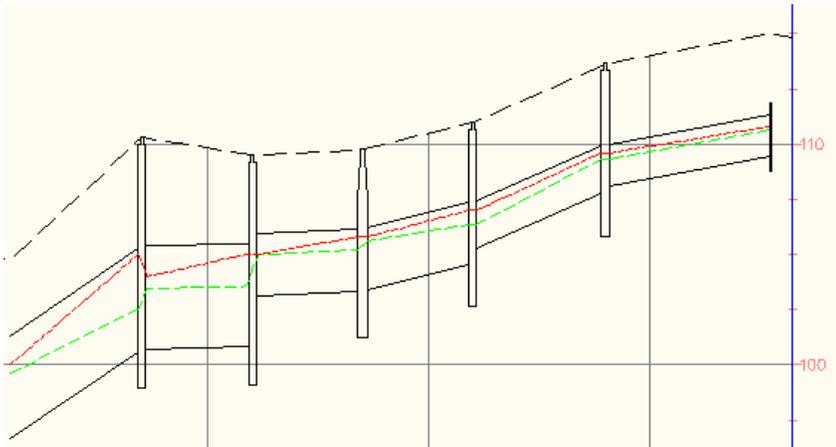


The energy grade line is displayed as a red dashed lined (on the top). The hydraulic grade line is displayed as a green dashed line (below the energy grade line).

In order to view hydraulic and energy grade lines in profile views, you must ensure that the following tasks have been performed:

| Task | Description |
|------|-------------|
| Add hydraulic property information to the pipe network by performing hydraulic analysis on the pipe network using the Storm Sewers Extension | For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280). |
| Edit the pipe style to configure display styles for the HGL and EGL components | Edit the HGL and EGL components listed on the Display tab of the Pipe Style dialog box. You only need to do this for pipe styles, not for structure styles. See Editing Pipe Display Styles for HGL and EGL Components (page 1283)for step-by-step instructions. |
| Create profile view and profile | For more information, see Displaying Pipe Networks in Profile Views (page 1220). |

You can also create label styles that display labels for hydraulic properties, such as HGL and EGL. For more information, see Labeling Hydraulic Properties in Pipe Networks (page 1270).

In addition to viewing the hydraulic and energy grade lines in profile views, you can also view and edit the hydraulic property data values in the following ways:

| Editing Method | Description |
|---|---|
| Pipe or Structure Properties dialog box | Click the Part Properties tab of the Pipe or Structure Properties dialog boxes, and scroll down to view the values for hydraulic properties. |
| Pipe Network Vista | Display the pipe network in the Pipe Network Vista, and scroll to view or edit the hydraulic property data values. |

For more information, see Displaying Pipe Networks in Profile Views (page 1220).

**To edit the pipe display style for HGL and EGL**

1   In Toolspace, on the Settings tab, expand the Pipe collection, then expand the Pipe Styles collection.

2   Double-click the desired pipe style to edit it.

3   Click the Display tab on the Pipe Style dialog box.

4   Select Profile in the View Direction drop-down.

5   For the Hydraulic Grade Line and Energy Grade Line components, turn on visibility, and set other display characteristics, such as color and layer, as desired.

# Editing Hydraulic Property Data

Editing a pipe network may cause the hydraulic property data to become invalid.

It is important to note that if you choose to edit a pipe network's physical properties after if has been imported from an *.stm* file, the hydraulic information can become invalid. To notify you of this possibility, a warning message similar to the following is displayed whenever you edit pipe network parts from a pipe network that has been imported from the Storm Sewers Extension:

**A pipe network containing hydraulic properties has been altered and the properties may be invalid.**

This message is displayed only the first time you try to edit the pipe network. If you try to edit the pipe network again, the message is not displayed.

If you do see this message, you may want to use the Edit in Storm Sewers command to export pipe network data and update hydraulic properties. As a minimum, it is recommended that you verify that the hydraulic information for the pipe network is still valid, and has not changed in such as way as to cause undesireable or unexpected results in your design.

If you are editing a pipe network that does not contain values for hydraulic properties, this message is not displayed.

# Understanding Event Viewer Messages

Event Viewer messages alert you to certain conditions when importing and exporting Storm Sewer data.

The following table provides information on some of the messages that can be displayed in the Event Viewer when importing or exporting Storm Sewer data.

| Event Viewer Message | Description |
|---|---|
| Multiple barrels were found in the *.stm* file. | The import is successful, however, one or more pipe networks in the *.stm* file con- |

| Event Viewer Message | Description |
|---|---|
| | tained a No. of Barrels property that is set to more than 1. In Storm Sewers a No. of Barrels property that is set to more than 1 creates two lines (pipes) with the same size, invert, slope, and so on, in parallel. AutoCAD Civil cannot represent pipes in this way. Therefore, when you attempt to import a Storm Sewers file that contains No. of Barrels set to more than 1, AutoCAD Civil ignores this value, and displays this message in the Event Viewer. |
| Error importing pipe network <pipe network name>: The current layer is locked. | You attempted to import pipe network data from an *.stm* file when the current layer in the drawing is locked, or the object layers for pipes or structures is locked. In this case, no new pipe networks are created, and no existing pipe networks are overwritten. The import is aborted. Unlock appropriate layers and try importing again. |
| Error overwriting pipe network <pipe network name>: Objects reside on a locked layer. | You attempted to update an existing pipe network with a pipe network from an *.stm* file when the current layer in the drawing is locked, or the object layers for pipes or structures is locked. In this case, no new pipe networks are created, and no existing pipe networks are overwritten. The import is aborted. Unlock appropriate layers and try importing again. |
| Pipe or structure <pipe or structure name> was not matched exactly. The closest larger matching part size was used. | The import was successful; however, one or more Storm Sewers parts could not find part sizes that matched exactly with Auto-CAD Civil part sizes. In cases like this, AutoCAD Civil uses the closest larger part size for the imported Storm Sewer part. |

# Pipe Network Command Reference

You can use commands to quickly access pipe network functionality.

The following table lists the pipe network AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
|---|---|
| AddNetworkLabels | Invokes a labeling tool to annotate pipes and structures (page 1261) |
| AddNetworkPartPlanLabel | Adds labels to selected pipes and structures in plan view (page 1261) |
| AddNetworkPartProfLabel | Adds labels to selected pipes and structures in profile view (page 1261) |

| Command | Description |
|---------|-------------|
| AddNetworkPartSectLabel | Adds labels to selected pipes and structures in section view (page 1265) |
| AddNetworkPartsToProf | Updates a profile view to show newly added pipe network parts (page 1220) |
| AddNetworkPipeTable | Creates a pipe network table (page 1271) |
| AddNetworkPlanLabels | Adds labels to a pipe network in plan view (page 1261) |
| AddNetworkProfLabels | Adds labels to a pipe network in profile view (page 1261) |
| AddNetworkSectLabels | Adds labels to a pipe network in section view (page 1261) |
| AddNetworkStructTable | Creates a pipe structure table (page 1272) |
| AddSpanningPipePlanLabel | Adds labels to connected pipe spans in plan view (page 1265) |
| AddSpanningPipeProfLabel | Adds labels to connected pipe spans in profile views (page 1265) |
| ApplyRules | Invokes a labeling tool to annotate pipes and structures (page 1250) |
| ChangeFlowDirection | Changes flow direction for selected pipes (page 1225) |
| CreateAlignFromNetwork | Creates an alignment from existing pipe network parts (page 1196) |
| CreateInterferenceCheck | Runs an interference check (page 1226) |
| CreateNetwork | Creates a pipe network by layout (page 1191) |
| CreateNetworkFromObject | Creates a pipe network from a polyline, line, arc, or feature line (page 1194) |
| CreateNetworkPartsList | Creates a pipe network parts list (page 1238) |
| CreateNetworkPartsListFull | Creates a parts list that contains all of the parts in the current pipe network catalog (page 1239) |
| CreateNetworkReference | Creates a pipe network reference (page 1199) |
| EditNetwork | Edits a pipe network (page 1203) |
| EditNetworkPartList | Edits a network part list (page 1241) |

| Command | Description |
| --- | --- |
| ExportStormSewerData | Exports pipe network data to an .stm file for future import into the Storm Sewers Extension (page 1279) |
| ImportStormSewerData | Imports pipe network data from the Storm Sewers Extension (.stm file) into AutoCAD Civil (page 1278) |
| PartBuilder | Invokes the part builder interface (page 1287) |
| ResetAnchorPipe | Resets the anchor position of a label that spans multiple pipe network parts (page 1270) |
| ResetNetworkLabels | Resets pipe network reference labels (page 1201) |
| SetNetworkCatalog | Sets a default path for pipe and structure catalogs (page 1243) |
| ShowSpanningPipes | Displays the span of pipes that are specified as a span for labeling a span of pipes with a single label (page 1265) |

# Understanding Part Builder

# 28

Part Builder lets you create and modify pipe network parts that are available in AutoCAD Civil part catalogs.

This section describes the fundamental concepts involved with using Part Builder and explores the user interface.

## Before You Begin

Part Builder is intended primarily for users who are experienced with parametric modeling design and with pipe network features.

Individuals who have experience with parametric modeling design, concepts, and techniques should be comfortable making use of the full spectrum of features available through Part Builder. Individuals with little or no experience with parametric modeling should be comfortable using the more basic Part Builder features, such as adding or changing part sizes in existing part families. For more information on these features, see Editing Part Family Sizes (page 1244).

Before using Part Builder, make sure you understand the following:

**WARNING** Before using Part Builder to create or modify pipe network parts, it is important that you create and save a backup copy of the AutoCAD Civil pipe network part catalogs and support files. These files are installed by default in: C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Pipes Catalog. Create and save a backup copy of this folder and its contents, including all subfolders and their corresponding content.

- Pipe parts created with Part Builder must have a width, a height, and a centerline. Therefore, any pipe shapes that do not meet this criteria are not supported in AutoCAD Civil pipe networks. This includes open channel pipes.

- The section profiles of a pipe must have an enclosed loop. Therefore, it is not possible to create pipe types such as horseshoe shape or U-shape (open channel).

- Pipe part definitions must include a uniform wall thickness. Therefore, you cannot create pipe parts with varying wall thicknesses.

- Part Builder cannot create a rectangular box culvert shape that has chamfers on the inner diameter corners.

- When a pipe connects to a structure, there is a single point of connection between the pipe and the structure. Therefore, you cannot customize a pipe part to have two connection points at one end of the pipe, where it connects to the structure.

For more information, see the Pipe Networks chapter in the AutoCAD Civil Best Practices Guide.

# Part Builder Overview

With Part Builder, you can design and edit shapes for drawing content that represent real-world pipe network parts, such as pipes, manholes, catch basins, and headwalls.

Part Builder is intended primarily for advanced users who are experienced with parametric modeling design and with pipe network features.

Each piece of content represents a part family, such as concrete pipe, ductile iron pipe, and concrete elliptical culvert. Each part family contains a collection of part sizes within that part family. For example, within the concrete pipe part family, there are a variety of pre-defined part sizes available for that part. Part Builder enables you to create and modify part families as well as individual part sizes.

When you use Part Builder, you can build 2D models of pipe parts, and/or 3D models of structure parts, and generate 2D drawing views of those parts to use in design layouts. The model you create is defined according to the size, shape, and position of the features that make up the part. Parts are stored in a part catalog, such as the US Imperial Pipe Catalog, and you can navigate to and select the part from within a catalog when you want to add it to your layout.

**Accessing Part Builder**

To access Part Builder, click Home tab ➤ Create Design panel ➤ 🔲 Part Builder.

When you do this, the Getting Started - Catalog Screen dialog box is displayed.

**Getting Started - Catalog Screen**

From the Getting Started - Catalog Screen dialog box, you can choose to work with the pipe catalog or structure catalog and navigate through the categories within that catalog.
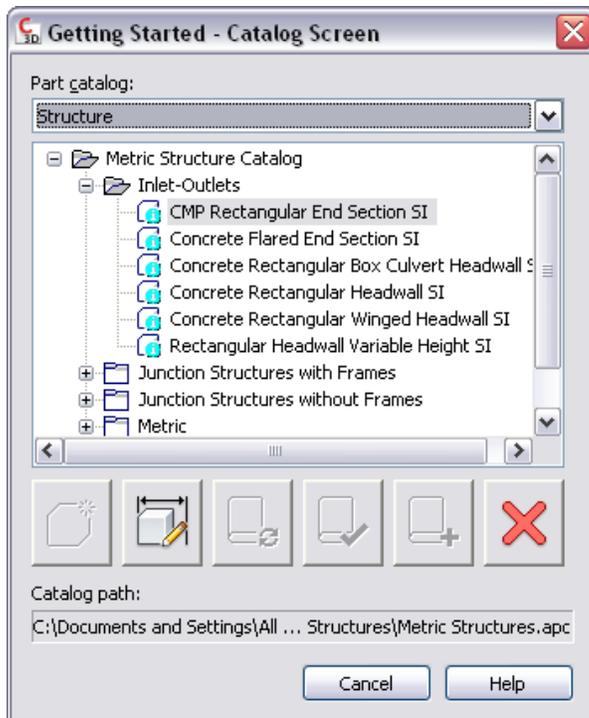
The icons on this dialog box let you perform several tasks. If you position your mouse over an icon, a tooltip provides an explanation of the icon. Icons are activated based on what is selected in the catalog tree.

| If you want to… | then do this … |
|---|---|
| create a new part | select a chapter folder then click New Parametric Part. Enter a name and description for the new part on the New Part dialog box, and click OK. After doing this, the building environment is opened for part creation. |
| modify an existing part size | select the part in the catalog tree, and click Modify Part Sizes. The building environment is opened for modifying the part. |
| regenerate the catalog | after making changes to parts or to a catalog, click Catalog Regen to regenerate the entire pipe or structure part catalog. |
| test (validate) the catalog | after making changes to a catalog, click Catalog Test to verify that the validation is successful. |
| add a new chapter for a part | click New Chapter and enter a name. You can add chapters to the catalog or to another chapter. |
| delete a part | select the part and click Delete. The part is deleted from the catalog and is removed from the catalog tree. |

**Parametric Building Environment**

When you choose to create or modify a part, Part Builder opens a parametric building environment that includes a variety of features:
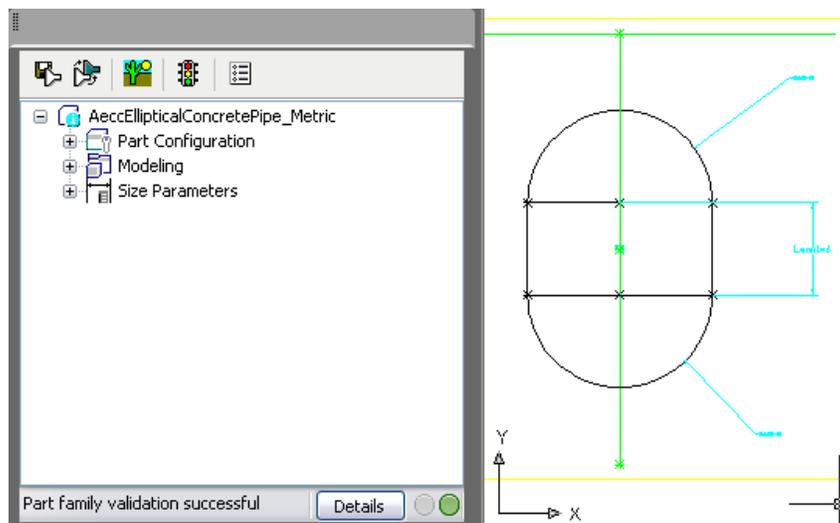


■ Part browser. The part browser is displayed on the left side of the screen. It provides an organized view into the features of the currently selected part. As you define your part, additional features are nested in a hierarchy under the appropriate folder. In the browser, you can show more or less detail by expanding or collapsing the folders.

- Modeling area (viewport). The modeling area (viewport) is displayed on the right side drawing space and is designed to give you full view control over your part model. You can use the standard AutoCAD view commands to change the direction in which you view your part model.

- Toolbar. The toolbar provides quick access to select Part Builder commands. Icons are available for saving your part, saving an existing part as a new part, generating a preview image, validating your part, and specifying part options.

- Status bar. The status bar, located at the bottom of the part browser, provides updated information about part validation when you click the Validate icon on the toolbar. A description of the validation status is provided. A details button provides a list of warnings and/or errors found in your model.

- Feature-specific menus. Feature-specific menus are available by right-clicking a part feature in the browser window. Options not available are shaded.

The part browser and the modeling area are resizable windows.

# Understanding Parametric Parts

Parametric parts give you the flexibility to create pipe network parts that are dynamically sized according to size parameters.

You define a parametric part by creating a 3D model of the part and assigning variables, or parameters, to control its shape and size. Parameters are defined by individual values, lists of values, or calculated values based on equations. When the pipe network part is placed in your drawing, you select specific part sizes and the part is dynamically built to reflect the specified shape and size. Parametric parts enable you to create a single 3D model for a part family that can be updated to represent multiple part sizes. You can define millions of parts through a single model, depending on the number and type of parameters defined.

It is beneficial to create pipe network shapes as parametric parts for many reasons. Many of the shapes used to represent pipe network components share the same geometric features; however, the features change in size according to the design. For example, an 8-inch circular PVC pipe looks just like 10-inch PVC pipe except for the pipe diameter. Similarly, you may have two headwall structures that are the same basic shape but have different heights, widths, or both.

- Pipes: An 8-inch circular pipe looks the same as a 10-inch circular pipe, except for the size of the diameter. By creating a parametric part, you can create a 2D cross section of a circular pipe and assign a parameter for the pipe diameter that can change in size. When the pipe is placed in your drawing, the 3D pipe uses this cross-sectional shape parametrically sized to match your part size selection. You specify which size pipe you want and the appropriate diameter is dynamically built to match.

- Structures: A 38-inch concentric catch basin looks the same as a 48-inch catch basin, except for the height and diameter. By creating a parametric part, you can create a 3D model of a catch basin structure and assign parameters for the height and diameter that can change in size. When the catch basin is placed in your drawing, you can specify the height and diameter, and the appropriate size catch basin structure is dynamically built to match.

# Key Terms

Understanding the following key terms will help as you begin using Part Builder.

**constraint** Controls the shape of a feature by establishing relationships between features in the model.

**degree of freedom** In part modeling, determines how a geometric object, such as a line, arc, or circle, can change shape or size. For example, a circle has two degrees of freedom: center and radius. When these values are fixed, degrees of freedom are said to be eliminated.

**dimension** Controls the size of a feature. When changed, the feature is resized. May be expressed as a constant value, a value as part of a table, a calculated value, or a list of values.

**domain** Refers to the part category. There are only two domain types by default: pipes and structures. Each domain type has a unique behavior in AutoCAD Civil.

**feature** A building block of a pipe network part model. You combine features to create pipe network part models.

**model** The part shape representing all possible sizes, and defined by parameters, geometry, dimensions, and geometric constraints.

**parametric** A solution method that uses the values of part parameters to dynamically size the part.

**part size** A specific set of values applied to the parametric model to define a single size.

**work plane** An infinite plane related to one or more features of the model. Work planes provide a defined place in space from which to build the model.

# Overview of Creating a Part

This section provides an overview of the steps involved with creating a part.

When you create a pipe network part, you define the features of the part and how they are associated with one another. Each pipe network part represents a unique part family. You should analyze the pipe or structure catalog hierarchy to determine the best place to add part families. It is also important to notice the relationships between the different part sizes you want to create. The features you define in the model determine the flexibility you have to create multiple part sizes. You should look at the entire part you want to create, decide how to break it down into simple shapes, and determine the relationships to establish between the different shapes.

Creating a part with Part Builder involves performing the following tasks in the order they are presented below:

**1** Specify the part configuration.

You work in the parametric building environment to create single pipe network parts. While you are working in this environment, only one part can exist in a drawing. The individual drawings are associated with a part catalog to build a library of parts. You specify the type and subtype of the part to establish basic part behavior. For more information, see Part Configuration (page 1292).

**2** Create a 2D or 3D model of the part.

During this phase of the process, you create models of 2D pipe parts, or 3D structure parts, and use them to dynamically generate 2D views of the part. You build your part from features that are combined to define the part. The model is defined according to the size, shape, and position of its features. You can restrict how the features of the parts fit together. To better conceptualize the size and shape of the part model, you define dimensions, constraints, and modifiers that determine how your part is built. For more information, see Part Modeling (page 1293), Dimensions (page 1296), Constraints (page 1297), and Modifiers (page 1299).

**3** Generate a preview image and define the insertion position of the part.

Once you have finalized the model, you generate a preview image for the part by taking a snapshot of the model. You also specify the placement position that you use to place the part in your drawing. For more information, see Generating a Preview Image of a Part (page 1317) and Defining Part Insertion Position (page 1317).

**4** Validate and save the part.

Once you have finalized the model, you must validate and save the part. Validation checks the model and detects any errors that make the part invalid. You must correct all errors in order to successfully save the part and add it to a drawing. For more information, see Validating and Saving a Part (page 1318).

# Part Configuration

Defining the part configuration is the first step in creating a part.

When the parametric building environment opens, the top folder in the part browser is the Part Configuration folder. The part configuration defines the characteristics and behavior of the part according to the domain (pipe or structure), type, and subtype, and it is required to save the part. To ensure that the part can be validated and saved correctly, you should define the part configuration before starting to model the part.
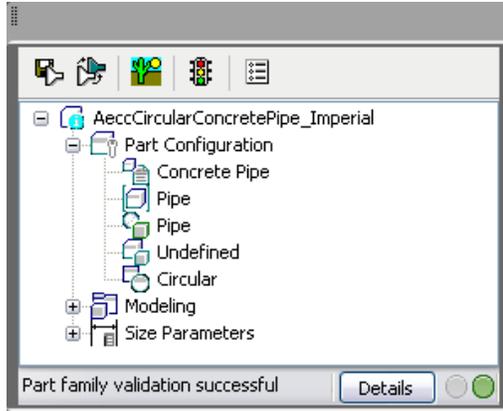


The part configuration is defined once for a part family and remains constant for all part sizes. The following components are required to define the part configuration:

**Part Name** You cannot change the part name of existing (default) parts. This is because part names for existing parts are used in the filenames for the parts. You can, however, create a new name for a new part using Save as.

**Part Description** Describes the part family. When you create and name a new part, you enter the description in the New Part dialog box. By default, the description is the same as the part name unless a different description is entered. For existing parts, the part description is predefined.

**Part Domain** Defines the family of parts. The two basic domains are pipes and structures. You cannot edit the part domain in the part browser. The part domain is predefined based on the part catalog you selected in the Getting Started - Catalog Screen dialog box of Part Builder. The part domain is selected from a list of predefined domains for pipe networks (pipes and structures). In the illustration above, the part domain is Structure.

**Part Type** The part type defines certain behavior properties that are assigned to the part.

For pipes, the predefined part type is always "pipe". A variety of pipe *shapes* may be available; for example, circular, egg-shaped, elliptical, and rectangular. However, they are all defined with same part type (*pipe*).

For structures, the predefined part types include general, inlet-outlet structures, and junction structures. As previously mentioned, the part type defines certain application behavior properties that are associated with the part. A general structure, for example, has a different set of application behavior properties than an inlet-outlet structure, and so on. In the illustration above, the part type is Inlet-Outlet Structure.

From the part browser, you can specify the part type by right-clicking on an item under Part Configuration and choosing Edit. The list of predefined part types is displayed.

**Part Subtype** Lets you organize parts into more detailed, logical groupings. This grouping does not affect the part behavior. In the part browser, you specify the part subtype from the list of predefined subtypes. You can also enter a custom part subtype if you wish. The part subtype is helpful during part selection to filter a large group of parts that are of a similar type. In the illustration above, the part subtype is Undefined.

# Part Modeling

Modeling a part involves configuring a variety of part features.

In Part Builder, the term model refers to the graphical representation of a part. A model consists of various features that have specific relationships to each other and that define the behavior of the part. Some features require that you create simple shapes or points, while others require an extrusion or path. Some features represent visible geometry, and some help you to position geometry precisely on a part. You can modify features to refine and improve your parts over time. You change features by modifying their size and shape or by flipping or moving them. To effectively model a part, it is important to understand how each feature relates to the other features of modeling.

The following illustration shows the features that make up the model. Arrows indicate the direction of the relationship between features. For example, modifying geometry affects a profile, which affects a modifier, and so on.
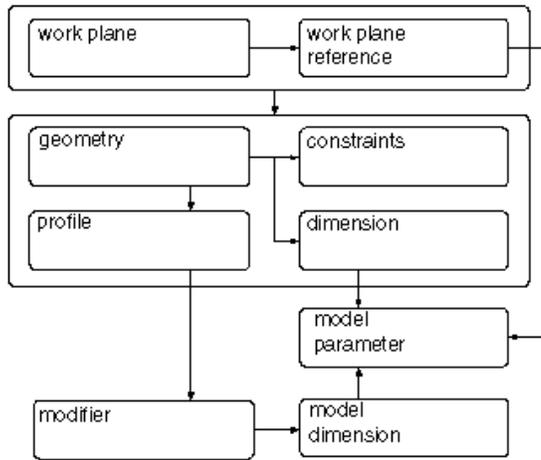


The following sections introduce the features of modeling. Refer to this illustration as you learn more about each feature to help you understand the relationships between features.

# Work Planes Overview

Use work planes to define a part, position geometry, and define relationships between part features.

In Part Builder, a work plane is a modeling feature that defines the location of a plane in three-dimensional (3D) space. It is an infinite construction plane that can be placed at any orientation in space. It can be offset from an existing work plane, or it can reference 3D geometry. Using a work plane, you define the geometry, dimensions, constraints, and profiles that make up the part model. Work planes help you to place geometry that would otherwise be difficult to position. By constraining geometry to work planes, you can control their location. Work planes help you to define relationships between features and provide control when placing features.

A work plane is displayed as a rectangular two-dimensional (2D) object. The work plane display is only a visual representation of the infinite plane and cannot be moved or resized. However, you can control its visibility for ease of viewing the model. Offset and reference work planes are user-defined and provide the flexibility to be moved and redefined.

**NOTE** To ensure a manageable model size, it is recommended to use a minimum number of work planes.

When you right-click on a work plane in the part browser, it is highlighted in the modeling area. You can change the view direction to match that of the selected work plane when adding geometry or dimensions by using the Set View option on the Work Plane shortcut menu.

**IMPORTANT** Any features attached to a work plane are restricted to the original plane. If you move a work plane, any features attached to the plane also move. If you delete a work plane, any features attached to the plane are also deleted. Each feature attached to a work plane appears under the Work Plane folder in the part browser.

Part Builder provides three default work planes that intersect at the origin of the *X*, *Y*, and *Z* axes. The default work planes help you to get started with modeling a part. Generally, it is best to start your modeling in the top work plane, and add others as needed. You can add work planes at any time during the modeling process. Each work plane has its own internal coordinate system. Work planes can be created on any plane in the current user coordinate system (UCS) or in the World Coordinate System (WCS).

The following preset work planes can be added to a model from the Create Work Plane dialog box:

**Default** Creates the standard *ZX*, *YZ*, and *XY* work planes of the WCS.

**Custom** Creates a work plane that is not available by the preset work planes. The work plane has user-defined values for the *X* and *Y* direction and the origin of the plane in the current UCS.

**Offset** Creates a work plane that is offset by a specified distance from a selected source work plane. For more information, see Offset and Reference Work Planes (page 1294).

**Reference** Creates a work plane that is attached to the extents of a modifier feature. For more information, see Offset and Reference Work Planes (page 1294).

**Top** Creates a work plane that matches the standard Top 3D view.

**Bottom** Creates a work plane that matches the standard Bottom 3D view.

**Front** Creates a work plane that matches the standard Front 3D view.

**Back** Creates a work plane that matches the standard Back 3D view.

**Left** Creates a work plane that matches the standard Left 3D view.

**Right** Creates a work plane that matches the standard Right 3D view.

**Viewing default work planes**



ZX work plane          YZ work plane          XY work plane
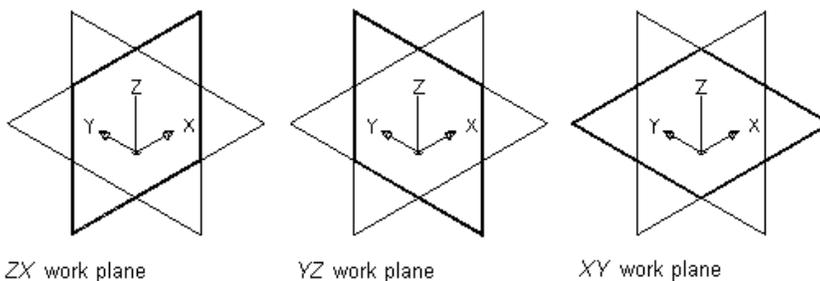
# Offset and Reference Work Planes

Offset and reference work planes are relational and are based on one or more defined features in the model, such as other work planes or modifiers.

**Offset Work Planes**

An offset work plane is a specified distance away from another work plane. This work plane can be offset from any existing work plane, including another offset or reference work plane. You define an offset work plane by selecting a source work plane and specifying a distance between the work planes. You can use offset work planes to maintain specified or calculated distances between features, such as profiles, geometry, or modifiers. You can also use offset work planes as construction guides for locating features that would otherwise be difficult to locate. For example, use an offset work plane to define the length of a transition.

**Creating an offset work plane**
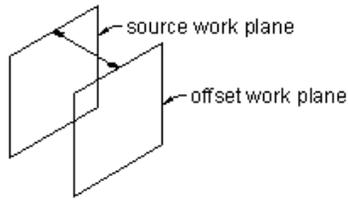


specify offset
distance

### Reference Work Planes

A reference work plane is defined as a plane on the face of the extents of a modifier. Every modifier has an invisible extent, or boundary box, that defines the extents of the feature. You can create a work plane that references one of the planes of the extent. To define a reference work plane, you select a modifier and a source work plane that represents the plane direction you want to create. The extents, or boundary box, of the feature is detected, and two valid reference work planes are available for selection. If the modifier is moved or resized, the reference work plane is moved with it.

**Creating a reference work plane**
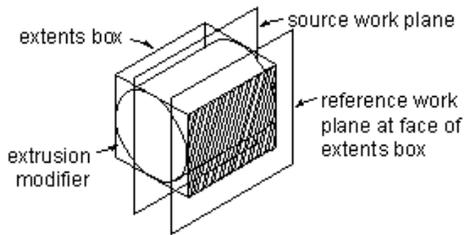


## Geometry

Geometry features are the basic building blocks for defining the size and shape of the model.

In Part Builder, geometry that you define is constraint based, two-dimensional (2D), and must be attached to a work plane. You can switch between work planes to define different geometry. As you add geometry to a work plane, the geometry features appear under the Geometry folder of the associated work plane.

---

**WARNING** Avoid using basic AutoCAD geometry commands. AutoCAD geometry is not valid for work planes and cannot be used to create features for part models.

---

**Viewing geometry types**
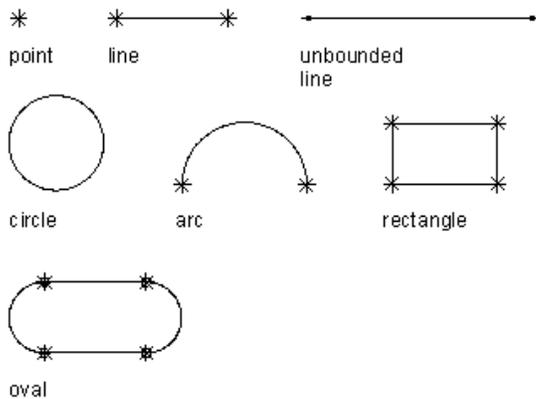


Part Builder provides nine types of geometry you can use to build the model:

**Point** Defined by an *X* and *Y* coordinate.

**Line** Defined by a position and a direction; constrained by a start point and endpoint.

**Unbounded Line** Defined by a position and a direction; infinite in length because it has no constraining start point or endpoint.

**Circle** Defined by a center point and a radius.

**Arc** Defined by a center point and a radius; constrained by a start point and endpoint.

**Rectangle** Defined by lines and points that are constrained to maintain start points and endpoints for each side and perpendicular angle.

**Oval** Defined by lines, arc, and points that are constrained to create two arcs tangent to two lines, with defined start points and endpoints.

**Point Reference** Defined by a point in the work plane that is based on a source point selected from a different work plane. A point reference is moved or deleted with the source point.

**Project Geometry** Defined by a projection of a modifier on a specified work plane. The project geometry is fixed and cannot be moved in the work plane. It is linked to the modifier and adjusts as the modifier changes.

# Dimensions

Add dimension information to specify the length, diameter, or rotation angle of geometric elements in the model.

Models require dimension information to define the size and position for the design. Dimensions are typically added after you have finalized the geometric features of the model; however, you can add dimensions at any time during the creation process. When you add dimensions, you apply rules that control the size and position of features in the model. Dimensions work in conjunction with constraints. The model is updated when changes are made to the dimensions.

Dimensions specify the length, diameter, or rotation angle of geometric elements in the model. When a dimension is added, a corresponding size parameter is also added. This parameter creates a placeholder value for the dimension that provides flexibility for defining the values of dimensions. Dimensions can be defined as default numeric constants or as equations. Although you can use them interchangeably, they each have specific uses.

- Numeric constants are useful when a geometric element has a static or fixed size that is populated as a value in a basic table, list, or constant storage type.

- Equations are useful when the size of a feature icon must be mathematically defined relative to the size of another feature.

Part Builder assigns a variable name to each dimension parameter. Letters and numbers are used to signify the type of dimension (such as length or diameter) and the sequence in which the dimension was added to the model (1 for first, 2 for second, and so on). To keep the model shape from becoming distorted as the dimensions resize it, define the large dimensions first. Dimension type depends on the feature you choose and where you place the dimensions.

**NOTE** When adding dimensions it is recommended to select points to specify the start and end locations. It is also helpful to turn off geometry that you not dimensioning for ease of selection.

**Viewing dimension types**



horizontal      vertical

parallel      perpendicular
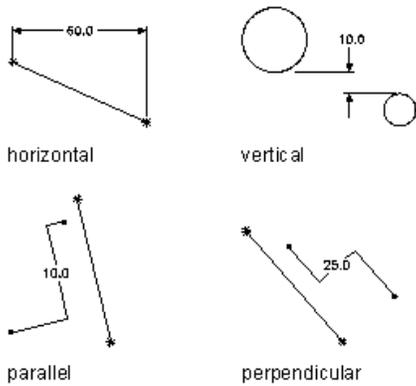
Part Builder provides seven types of dimensions you can add to the model:

**Distance** Can be defined for pairs of geometry of all types. Defines a value between two features. The value of a distance dimension cannot be negative. When adding distance dimensions between two lines, it is implied that the lines are parallel with a given separation.

**Horizontal Distance** Can be defined for pairs of geometry of all types. Defines a value between two features in a horizontal direction. The value of a distance dimension cannot be negative.

**Vertical Distance** Can be defined for pairs of geometry of all types. Defines a value between two features in a vertical direction. The value of a distance dimension cannot be negative.

**Parallel Distance** Can be defined for pairs of geometry of all types. Defines a value between two features in a parallel direction. The value of a distance dimension cannot be negative.

**Perpendicular Distance** Can be defined for pairs of geometry of all types. Defines a value between two features in a perpendicular direction. The value of a distance dimension cannot be negative.

**Diameter** Can be defined for circular and arc geometry. Defines a value for the diameter of a circular feature.

**Angle** Can be defined for pairs of linear geometry. Defines a value for the degrees between two linear features.

# Constraints

Add constraints to create rules that control how a part can change in shape or size.

Depending on the geometry of the model, you may need to add one or more constraints to define the shape or size of the model. Constraints enforce rules that you want the model to obey. Constraining a model controls how a model can change in shape or size, called "degrees of freedom." For example, a circle has two degrees of freedom: the location of its center and its diameter. If the center and diameter are defined, the circle is fully constrained and those values can be maintained. Constraints specify the relationships of geometric features; for example, whether two features are perpendicular, parallel, tangent, concentric, or have the same midpoint or radius.

**NOTE** After you add geometry, dimensions, or constraints to the model, the degrees of freedom are listed on the command line.

Constraints work in conjunction with dimensions to control the shape and size of the model. Any time you modify the model, the geometry retains the relationships among features in accordance with the applied constraints. You add constraints to indicate your design intent. For example, a parallel constraint could be defined between two lines based on the geometry of the model. You could also add a constraint to force both lines to have the same length.

As you apply geometric constraints, continue to analyze the model, reviewing and replacing constraints as necessary. As you gain experience, you will be able to determine which constraints control the model to

meet your design requirements. Some constraints work only with lines, while others work only with arcs, circles, or points.

See Adding a Constraint (page 1315) for step by step instructions on how to add a constraint.

**Applying a tangent constraint**



select first geometry

select second geometry

resulting added tangent constraint
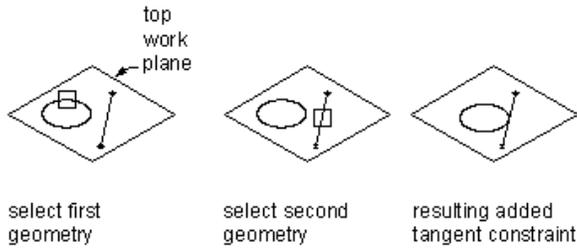
**Applying a parallel constraint**



select first geometry

select second geometry

resulting added parallel constraint
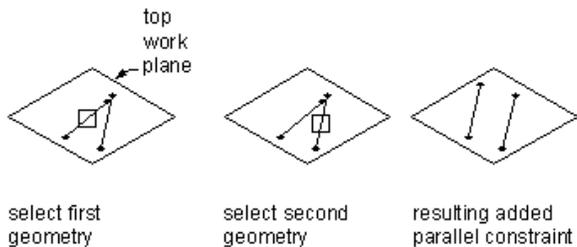
**Applying a perpendicular constraint**



select first geometry

select second geometry

resulting added perpendicular constraint
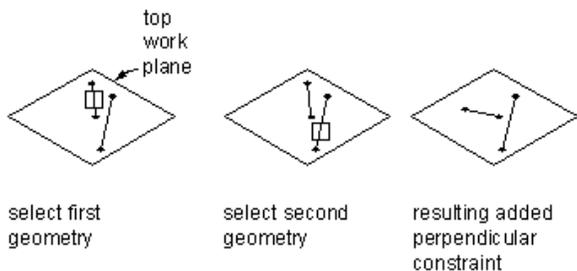
**Applying a concentric constraint**



select first geometry

select second geometry

resulting added concentric constraint
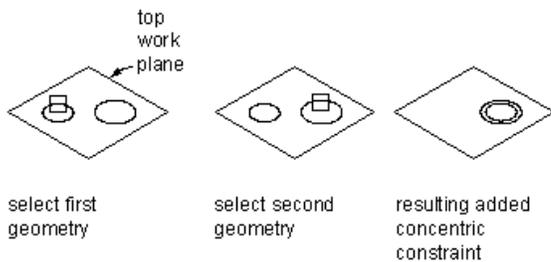
Part Builder provides ten geometric constraints. The following list describes these constraints and the features with which they can be used.

**Tangent** Can be defined between curved geometry (such as a circle or arc) and either another curved geometry or a line. Makes two curves tangential to one another, even if they do not physically share a point. Tangency is commonly used to constrain a line to an arc or circle.

**Parallel** Can be defined between pairs of geometry with a direction, such as lines. Causes two or more lines to be parallel to one another.

**Perpendicular** Can be defined between pairs of geometry with a direction, such as lines. Causes selected lines to lie at right angles to one another.

**Concentric** Can be defined for any combination of circles and points. Fixes the centers of the geometry to the same location. Common uses include circle to circle, where the center of both circles is the same; circle to point, where the point lies at the center of the circle; and point to point, where the points are the same.

**Coincident** Can be defined between a point and any geometry. Fixes two points (including center points) together; essentially, the point lies on the geometry.

**Equal Distance** Can be defined between two pairs of geometry. The distance between the first pair of geometries is fixed to the distance between the second pair. Equal distance constraints do not control the actual distance. Each pair of geometries must be one of the following: any combination of points and lines, two circles or arcs concentrically constrained, or a point and circle or arc concentrically constrained.

**Equal Radius** Can be defined between two circles or two arcs. Fixes the radius of both circles or arcs to be of the same value. Equal radius constraints do not control the value of the radii.

**Midpoint** Can be defined between a point and either two other points or two lines. The point is equal distance from the other two geometries. Midpoint constraints do not control the distance. A common use is constraining a point to the middle of a line.

**Symmetric** Can be defined between two geometries of the same type and a line. The two geometries are symmetrically arranged on opposite sides of the line. The symmetric constraint does not force constrained geometry to maintain an exact mirror image.

**Normal** Can be defined between a line or curve and a curve. (Two lines cannot be made normal; a perpendicular constraint must be used instead.) The curves intersect and the directions of curve tangents are perpendicular at the point of intersection. A common use is constraining a line to the normal of an ellipse.

# Profiles

Create profiles to define a two-dimensional (2D) outline of a part's geometric shape.

Using Part Builder, creating a profile is as easy as drawing a closed shape. Profiles are similar to geometry in that they are a visual representation of the 2D shapes that make up the model. Because profiles automatically associate constraints to the geometry, you can use profiles as a source of information from which to create features. You create profiles on a work plane and apply modifiers, such as extrusions, to them.

> **NOTE** Profiles can be used to create solids using modifiers. Non-profile geometry cannot be used to create solids directly. However, you can group a set of non-profile geometries together to create a custom profile.

Part Builder provides four types of profiles to use for creating features in the model:

**Circular** Creates a profile based on a circle defined by a center point and diameter to maintain its shape.

**Rectangular** Creates a profile based on a rectangle defined by four lines, four points, and four perpendicular constraints to maintain its shape.

**Oval** Creates a profile based on an oval defined by two lines, two arcs, four points, and four tangent constraints to maintain its shape.

**Custom** Creates a profile from existing geometry in the model. The geometry must be attached to a single work plane. Constraints are assigned as needed to maintain the shape of the geometry as it was selected.

# Modifiers

Use modifiers to create three-dimensional features of the model.

A modifier is a general term for any operation that affects features of the model. After creating profiles, you can use modifiers to extrude, add, or subtract features and to sweep a profile along a path. You can also use modifiers to create cut planes and transition features.

Part Builder provides six types of modifiers: extrusion, path, transition, cut plane, Boolean add, and Boolean subtract. The extrusion modifier is the most common type used in part modeling and is generally the base feature for a model. When you extrude a profile to create a feature, you specify how the feature will modify the shape by choosing one of four operations: midplane, plane, from-to, or blind.

See for step by step instructions on how to apply a modifer.

**Midplane Extrusion Modifier**

A midplane extrusion uses a profile as the center of the extrusion and sweeps the profile an equal distance away from each side of the center.

**Applying a midplane extrusion modifier**
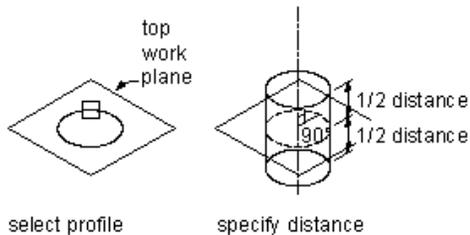


**Plane Extrusion Modifier**

A plane extrusion sweeps a profile between the profile itself and a specified work plane. If the work plane is a reference work plane, the extrusion is updated when the work plane is moved.

**Applying a plane extrusion modifier**
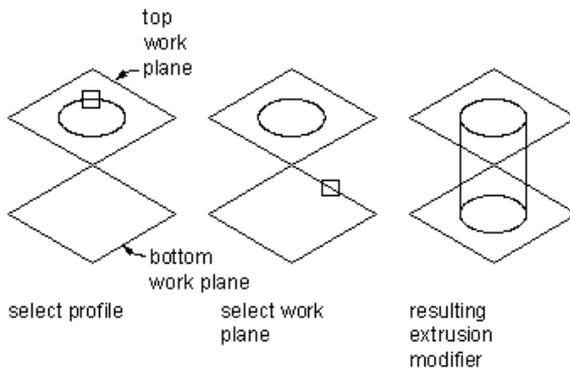


**From-To Extrusion Modifier**

A from-to extrusion sweeps a profile between two work planes.

**Applying a from-to extrusion modifier**



select profile     select to and from work planes     resulting extrusion modifier
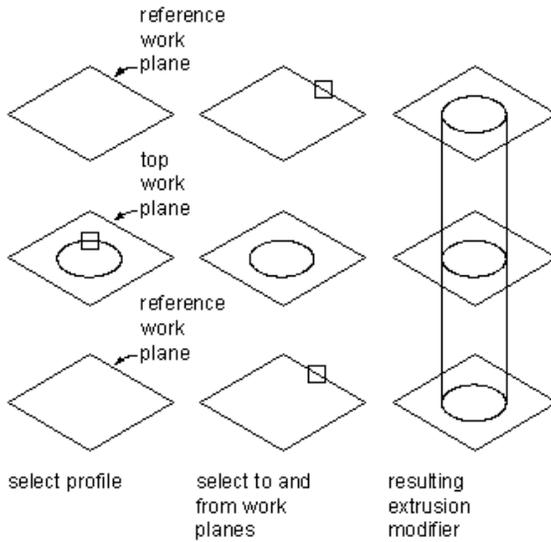
### Blind Extrusion Modifier

A blind extrusion sweeps a profile a specified distance along its normal, or perpendicular axis. You can flip the extrusion to sweep the profile in the opposite direction as the default normal.

**Applying a blind extrusion modifier**



select profile     specify distance     flipped extrusion modifier
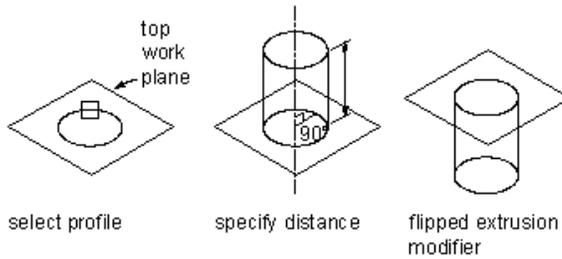
### Path Modifier

A path modifier sweeps a profile along path geometry. It creates a 3D feature based on the start and end profile of a piece of geometry, such as a line or an arc. The start and end profiles can be different.

**Applying a path modifier**



select path     select start profile

select end profile     resulting path modifier
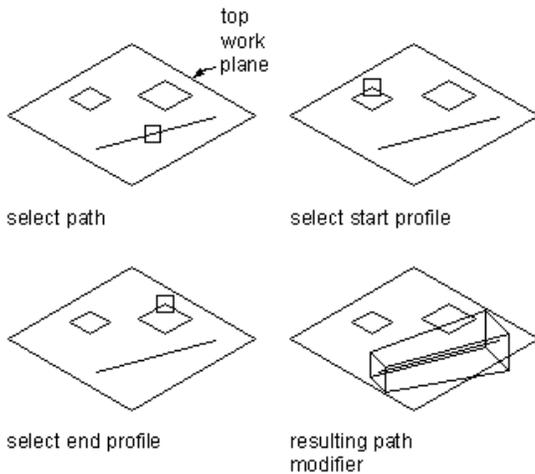
You can also specify the number of segments to use to create a 3D feature.

**Transition Modifier**

A transition modifier creates a transition body between two profiles. A transition includes both a start and end profile, which cannot be in the same work plane. Point references are useful to ensure that the profiles align between work planes; however, the centers do not need to align.

**Applying a transition modifier**
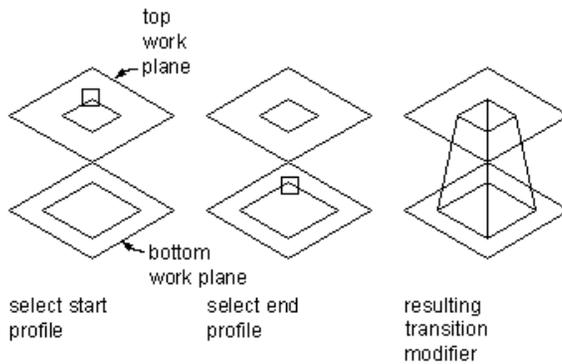


**Cut Plane Modifier**

A cut plane modifier cuts a feature into two parts, only one of which is kept. A cut plane modifier enables you to change the end of a feature to a slope, providing for the creation of more advanced features, such as an exhaust vent. You can modify the cut plane by dragging the normal, or perpendicular axis, of the modifier to create custom slopes. Cut planes are updated when the length of the modifier changes.

---

**NOTE** Cut planes cannot be defined parametrically. How they are defined in the model is how they will be displayed when placed in your drawing.

---

**Applying a cut plane modifier**
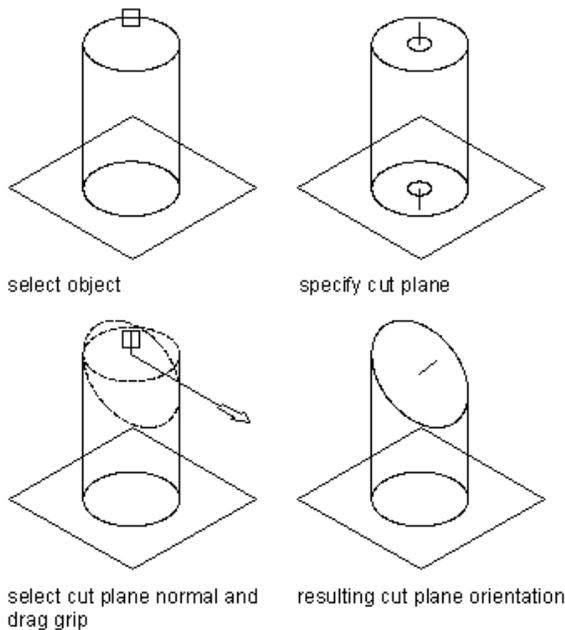
**Boolean Add Modifier**

A Boolean add modifier combines two features to create a single feature. This modifier assumes a basic concept of addition: that geometry inside other geometry, when added together, is removed. This modifier is helpful when creating models of parts that are placed in your drawing exactly as they were created in the model.

**Applying a Boolean add modifier**
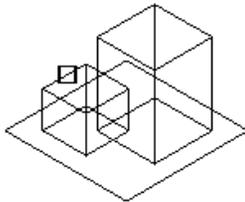


select first object        select second object        resulting Boolean addition modifier
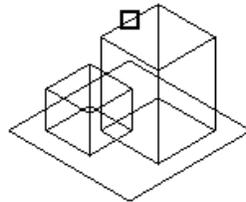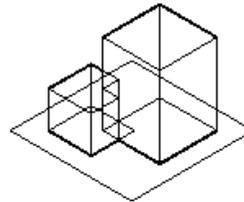
**Boolean Subtract Modifier**

A Boolean subtract modifier subtracts one or more features from another to create a new feature. The Boolean subtract modifier uses a basic formula when creating the resulting feature: that the subtractor objects are removed from the base object. To see results in the model, the subtractor features must intersect with the base feature. When no intersection of features exists, the subtractor features are removed from the base feature with no visible change in the model.

**Applying a Boolean subtract modifier**
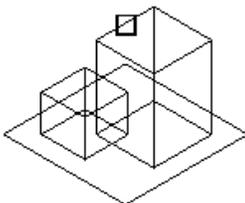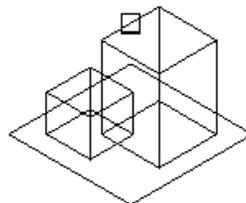


select base object        select subtractor object        resulting Boolean subtraction modifier
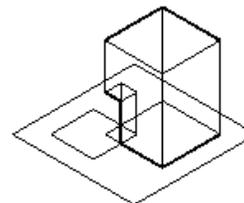
# Placement Points

Use the Autolayout Data option to define the part placement point in the drawing.

After finalizing the part model, you need to define the placement point for the part to ensure that it can be placed correctly in a drawing. The Autolayout Data option in the part browser controls how placement points can be defined. When you enable this option, an Autolayout Data folder is available in the part browser that includes commands for adding placement points.

This is done by selecting a fixed point in the model. This point must be in the z = 0 plane. By choosing a fixed point, the parametric model will retain this position as it is dynamically sized.
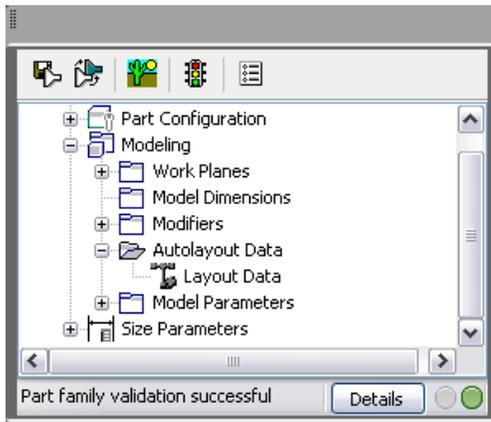
The placement point is the insertion point used to place the part in a drawing. You simply specify a point in your model for the placement point.

# Model Parameters

Configuring model parameters enables you to control the model's shape and size by establishing relationships between defined parameters.

As you add dimensions and constraints to the model, parameters appear under the Model Parameters folder in the part browser. The Model Parameters dialog box provides a central location where you can view, create, and edit the dimensional parameters of the model. You can define parameters to create additional model relationships later.

It is often more efficient to assign numeric values to dimensions and other feature parameters as you build the model. When you finalize the geometry, you can edit the model parameters to add calculated values. An equation assistant is available in the Model Parameters dialog box to ensure that the equation you create is valid and can produce a result. Each parameter in the Model Parameters dialog box includes the following:

**Name** A unique identifier for the model parameter. Model parameters are assigned default names, such as LenB1, (Length Body 1), WTh (Wall Thickness), PID (Pipe Inner Diameter), and BdyD1 (Body DImension 1), that can be changed. Model Parameter names are typically short acronyms because they can be used in equations. User-defined parameters must have unique names.

**Value** The result of evaluating the equation.

**Equation** The mathematical expression that defines the value of the parameter. Constants, other model and user-defined parameters, and mathematical operators can be used in an equation. You can use the Equation Assistant to define the equation.

**Description** A narrative, user-defined description of the parameter.

**NOTE** When using a constant value in an equation, be sure to define the constant as a model parameter. You should verify that the result of the equation is non-zero to avoid undesirable results in your model.

# Size Parameters

Part sizes are controlled by size parameters that set the dimensions of the part size.

Model parameters define the default part size of the model, and act as placeholders for new values that can be specified later. There is a direct connection between the model parameters and size parameters in that each model parameter is added to the size parameters in the part browser. Size parameters enable you to create different sizes for the model to represent multiple part sizes and to add non-graphical parameters to the part family.

You work with size parameters in the Edit Part Sizes dialog box. This dialog box lets you view and edit parameter values and configuration attributes in a table. Each parameter is viewed or modified independently of the others, so you can modify a single parameter at a time without scrolling through a large table.

The Edit Part Sizes dialog box has three working modes: Calculations, Parameter Configuration, and Values.

■ The Calculations mode provides access to the calculation formula of the parameter.

■ The Parameter Configuration mode provides access to attributes of the parameters, and includes description, data storage, data type, units, visibility, context, and index.

**NOTE** When parameters are added to the model, the name, data storage, context, and type attributes are assigned, and they cannot be changed.

■ The Values mode provides access to defined values of the parameter when they are stored as constants, lists, and tables.

**NOTE** Any calculations set in the model are fixed and can be changed only in the model.

The parameter configuration allows you to configure data storage. The type of data storage lets you define the parameter as a list or table of values, in addition to a constant or calculated value. You can also revise the parameter description and control parameter visibility. For more information, see Adding Individual Part Sizes (page 1312). You can define constant values, and copy and paste lists of values from other parts to define multiple part sizes. Calculated values can only be viewed. Use the Model Parameters dialog box to modify formulas for calculated values.

By creating a new parameter, you can define values that store additional information about the part. You can add custom parameters in which you define all parameter attributes, such as data storage type. You can also add parameters by selecting from a list of optional parameters that have been defined for you. The list of predefined parameters is determined by the part type and existing parameters found in the part family. These parameters provide additional part information that can help to identify parts in your drawing during the design process or when you produce a set of construction documents.

# Creating Parts with Part Builder

# 29

This section provides step-by-step procedures, illustrations, and tips for creating pipe network content using Part Builder.

In AutoCAD Civil, content is used to represent real-world pipe network parts such as pipes, manholes, catch basins, and headwalls. Pipe shapes are organized into the following part family categories based on the shape of the pipe: circular, egg-shaped, elliptical, or rectangular pipes. Structure shapes, such as manholes, catch basins, and headwalls, are organized into the following categories, or part families, based on shape characteristics: general structures, inlet-outlet structures (headwalls), junction structures (catch basins). Part Builder enables you to create and modify part families as well as individual parts.

## Tips for Creating Parts

Make sure you understand these tips before modeling parts with Part Builder.

### Tips for Using Part Builder

Keep the following tips in mind as you use Part Builder:

■ Save a backup copy of the part catalogs before using Part Builder, in case you need to revert to the original catalogs provided with AutoCAD Civil. You can use a browser application, such as Windows® Explorer, to copy and paste the catalogs and their sub-folders to a new location.

■ Determine model dependencies. Analyze the model design to determine how features interrelate; then decide how to create the model.

■ Work in a three-dimensional (3D) view. Creating the model in a two-dimensional (2D) view may lead to confusion.

■ Start any new model in the top work plane and pick your fixed insertion position before modeling.

■ Use the order of the folders in the part browser as a guide to the steps involved in the creation process.

■ Do not use the EXPLODE command. Exploding a part deletes the part definition from the catalog.

■ Part Builder generates drawing views of your pipe network part. The AutoCAD MVIEW command does not create associative views of your parts.

■ AutoCAD object snaps can be used to assist in object selection.

**Tips for Modeling the Part**

Keep the following tips in mind as you model the part:

- Use work planes to control the UCS orientation. Using the AutoCAD UCS command does not associate the current plane with your part.

- Use the AutoCAD Point Style command to increase point sizes. It is recommended to use an absolute point size.

- Use a minimum number of points. Reusing points is less confusing and helps in constraining the model.

- Use both constraints and dimensions. Some constraint combinations may distort unconstrained features of the model. If so, delete the last constraint and consider using a dimension or a different constraint combination.

- Use Part Builder dimensions. AutoCAD dimensions are not parametric and therefore cannot control the size, shape, or position of part content.

- Dimension large features before small features. To minimize distortion, define larger features that have an overall bearing on the model. Dimensioning small features first may restrict overall size. Delete or undo a dimension if the model shape is distorted.

- Define shape before size. If you apply constraints before dimensions, your model shape is less likely to become distorted.

- Determine model dependencies before applying constraints. A single constraint can often be used to define more than one feature's shape. Therefore, analyze the model design to help identify necessary constraints.

- Add constraints as needed to define the model shape. Because constraints often restrict more than one feature, use fewer constraints to avoid distorted models.

- When using calculated values for model parameters, be aware that calculated values are not available in the part Add Size dialog box.

# Creating a Part

This section contains step-by-step instructions for creating a simple part using Part Builder.

Each section describes a separate phase in the creation process. You should perform the steps in the order in which they are presented to avoid creating unusable parts.

# Creating a New Part

Use this procedure to start Part Builder and to begin creating a new part.

1 Click Home tab ➤ Create Design panel ➤ ▼ ➤ Part Builder 🔲 .
   The Getting Started - Catalog Screen dialog box is displayed.

2 Select the Structure part catalog by selecting Structure from the Part Catalog list at the top of this dialog box.

3 Select the Simple Shapes folder, then click New Parametric Part 🗋 .
   The New Part dialog box is displayed.

4 Enter a name for this new part family. For example, enter Simple Cylinder 2.

5 Click in the Description field.

By default, when you click in the Description field, the description is the same as the part name until you change it. You can type over this text to change it or add to it.

**6** Click OK to create the new, undefined part family.

The new part family is displayed in the part browser window.

**7** Proceed to Specifying the Part Configuration (page 1309).

## Specifying the Part Configuration

Use this procedure to specify the part configuration, defining certain behavior of the new part family you just created (Simple Cylinder 2).

**1** In the part browser, expand Part Configuration.

**2** To specify the part type, right-click Undefined Part Type, click Edit, and select a part type, such as General Structure.

The predefined part types are dependent on the selected part domain. For example, for a pipe part, the only available part type choice is "Pipe". For structure shapes, which are more complex than pipe shapes, there are multiple part type choices, such as general, inlet-outlet structures, junctions structures, and so on.

**3** To specify the part subtype, right-click Undefined Bounded Shape, click Edit, and select Cylinder.

The list of predefined subtypes depends on the selected type. For example for pipes, choices are arched, circular, egg-shaped, elliptical, rectangular, and undefined shape. For structures, you can specify the following subtypes for undefined bounded shapes: box, cylinder, sphere, and undefined.

**4** If desired, you can also enter a new subtype.

**5** Proceed to Modeling a Part (page 1309).

## Modeling a Part

To model a part, you must first define work planes on which to create the geometry of your model.

You can then create profiles and apply modifiers to define the shape and default size of the part. To avoid distortion in the model, start with the larger features, which have more impact on the overall size of the part, and then add the smaller features.

## Establishing Work Planes

Use this procedure to add work planes on which to create the geometry of the model.

To avoid confusion, work with only one work plane visible, unless you are using offset or reference work planes.

**1** In the part browser, expand Modeling.

**2** Right-click Work Planes and click Add Work Plane.

The Create Work Plane dialog box is displayed.

**3** Click Top and then click OK.

The top work plane is created.

You can add other types of work planes as needed. For more information, see Work Planes Overview (page 1293).

**4** To view the work planes in the modeling area, on the View menu, click 3D Views ➤ SW Isometric.

You can select any view direction or use the Views toolbar to zoom extents of your work planes.

**5** In the part browser, expand the Work Planes folder.

When you select a work plane in the part browser, the work plane is highlighted in the modeling area.

**6** Add more work planes as necessary to model the shape.

To avoid confusion, work with only one work plane visible at a time. To make only one work plane visible at a time, in the part browser, right-click a work plane and remove the check from the Visible check box (uncheck it).

When a workplane is marked as unvisible, the workplane icon in the part browser is grayed out, and the work plane itself is not displayed in the modeling area.

## Modeling a New Part

Use this procedure to model the new part you are adding.

Modeling any part typically involves adding a profile, geometry, dimensions, and constraints. You may need to add extrusions, but these are typically used for more complex structure shapes.

For a simple cylindrical structure part, the profile is circular, the geometry is typically fixed center, and an example of a constraint is concentric. Dimension parameters define various diameters for the shape. For example, a dimension parameter named BdyD1 is used to define the diameter for the body of a simple cylindrical structure.

**1** To create the new cylindrical structure, in the part browser, right-click XY Plane and click Add Profile ➤ Circular.

For more information about profiles, see Profiles (page 1299).

**2** Select a center point in the modeling area and specify a second point, or enter a value such as 12, to define the cylindrical shape radius.
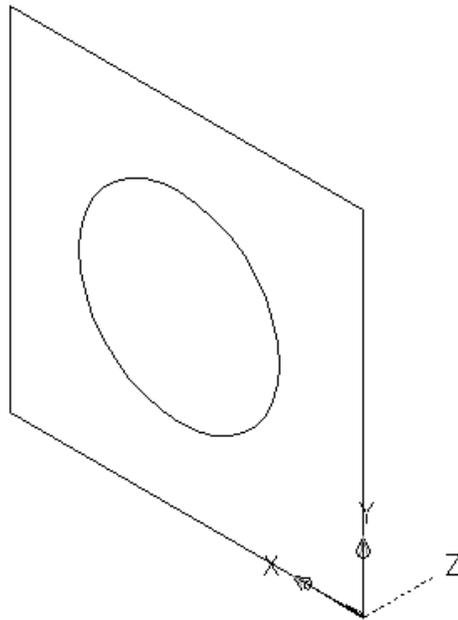
A circular profile is created and is added to XY Plane in the part browser.

**3** The next step is to add model dimensions to the part. Proceed to Adding Model Dimensions (page 1311).

For some shapes, you may want to add constraints or extrusion modifiers. For more information, see Constraints and Modifiers in Understanding Part Builder.

## Adding Model Dimensions

Use this procedure to add dimensions to specify the overall default size of the model.

Dimensions are used to define the default size of a part family (or part within a part family). You can add other sizes, such as a list of values, to create individual part sizes within a part family.

**TIP** You may find it easier to add model dimensions in the top view, rather than in a model view direction. To change the view, on the View menu, click 3D Views ➤ Top.

**1** To define the overall length of the part, in the part browser, right-click Model Dimensions, and then click Add Distance.

**NOTE** For constant parameter values, you can define a constant dimension, or choose not to add a dimension and use the default value based on the actual size of the geometry. When no dimensions are added, the parameter and its value are not displayed during part size selection.

**2** Select the model modifier feature in the modeling area, and specify a location to place the dimension.

A length dimension (for example, LenB1 or LenB2) is placed in the model and is added to Model Parameters and Size Parameters in the part browser.

**3** To modify the default, or to specify a calculated value for the length of the component, in the part browser, expand Model Parameters, right-click a length dimension, and click Edit.

The Model Parameters dialog box is displayed.

**4** Double-click the Equation column value for a model parameter (such as LenB1), and you can enter a value or an equation to specify the length of the part.

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**TIP** Click Calculator to access the Equation Assistant.

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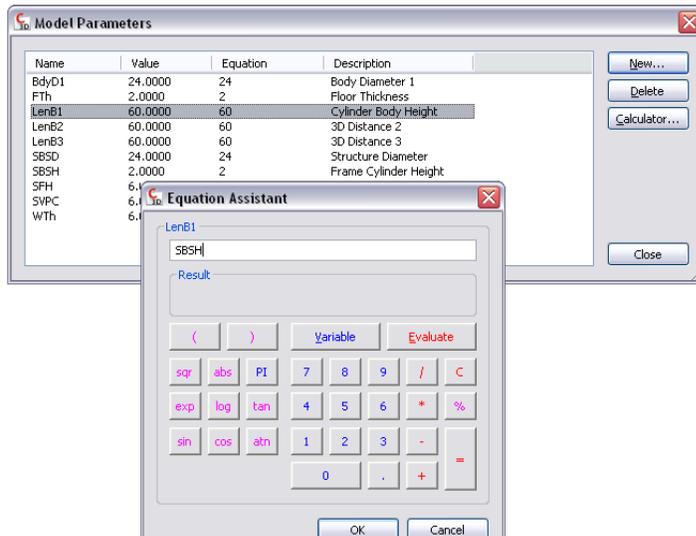


**5** The next step is to add individual part sizes to the model. Proceed to Adding Individual Part Sizes (page 1312)

## Adding Individual Part Sizes

Use this procedure to add individual part sizes to the model.

You create individual part sizes by adding parameter values. Values can be a list or table of values, a constant value, or a calculation.

You can also create unique part size names. Each part size name is generated using a calculation – a formatted string of parameter values and text.

The set of default size parameters that are available for each part differs depending on the part type. For example, a structure part defined as a junction structure (part type = junction structure) has a certain set of default size parameters that are appropriate for junction structures. Inlet-outlet structures (part type = inlet-outlet structure) have a different set of default size parameters available. Pipe parts have yet another set of available size parameters.

**1** To add part sizes, in the part browser, right-click Size Parameters and click Edit Configuration.

The Edit Part Sizes dialog box is displayed, showing all of the currently available size parameters for the selected part. For the Simple Cylindrical Structure part example, some default size parameters are Part Size Name (PrtSN) and Structure Vertical Pipe Clearance (SVPC).
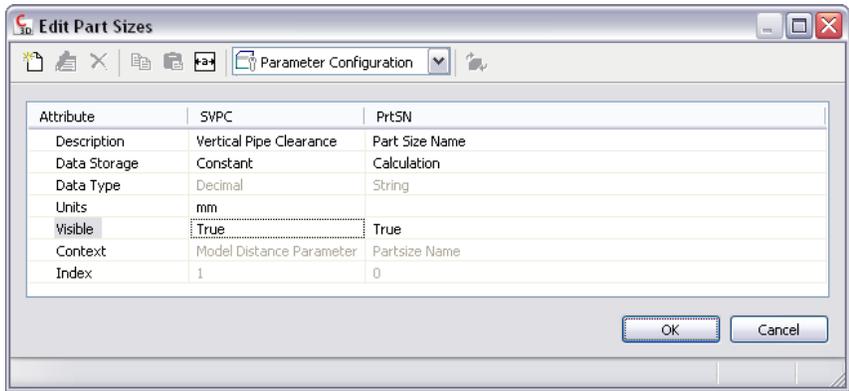
**NOTE** Calculated values set in the model cannot be edited in the Edit Part Sizes dialog box. To change these values you must edit the model.

**2** To add a list of available sizes for the part, change the data storage type of a size parameter to List.
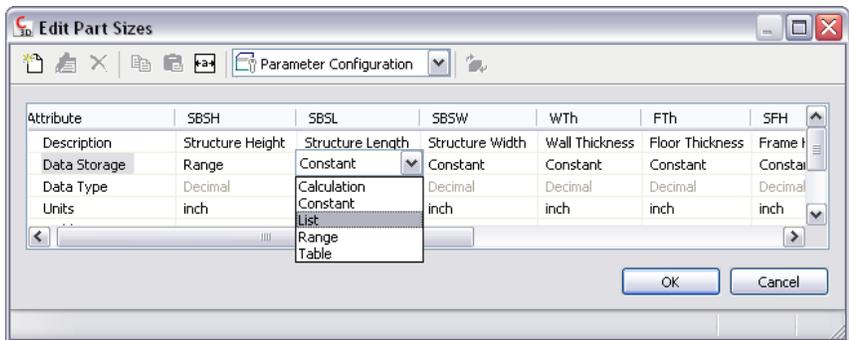


**3** In the toolbar, select Values from the list.

The parameter values are displayed.
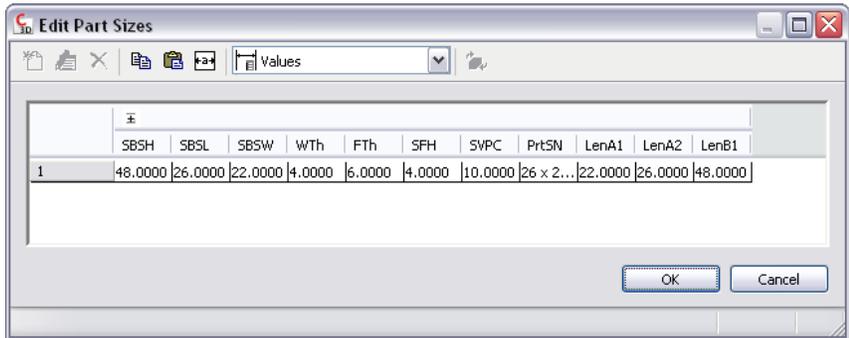


**4** To add new sizes for the part, click a parameter value such as the LenB1 parameter, and click Edit on the toolbar.

The Edit Values dialog box is displayed.
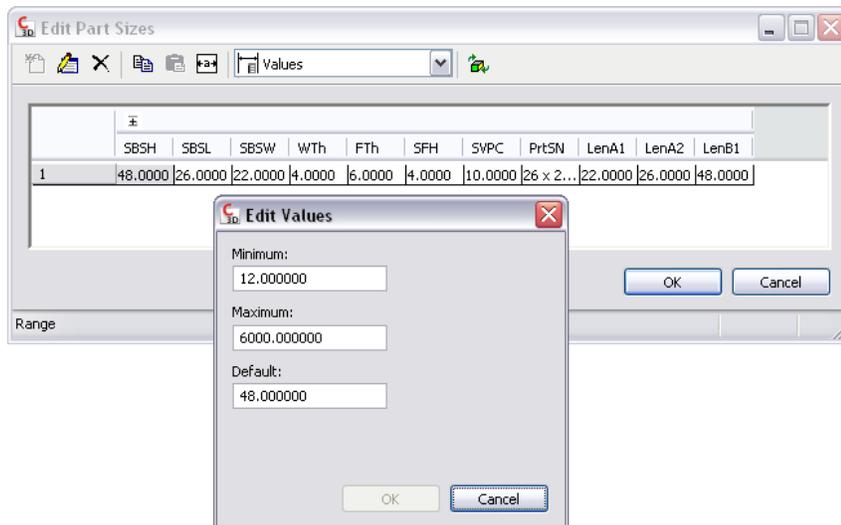
**5** Click Add and enter a new size. When you are finished adding sizes, click OK.

> **TIP** You can cut and paste values from other part families using standard Microsoft® Windows cut and paste functionality (CTRL+C and CTRL+V). Open another part in Part Builder, select the size parameter you want to copy, and then paste the selected values in the desired parameter of your part. You can also create a list of values in Microsoft® Excel, select the list of values you want to add, and then paste the values in the desired parameter of your part.

Now when you click in the size parameter value you edited (for example, LenB1) on the Edit Part Sizes dialog box, the list of sizes you just added is displayed.

**6** To specify a unique, calculated part size name, select Calculations from the toolbar list box.

The calculation strings associated with parameter values are displayed. For example, the calculation formula for a Part Size Name (PrtSN) for a structure could be "SHBW x SHBTh x SBSH inch Concrete Rectangular Headwall". Or for a pipe, it could be FormatNumber($PID,0) + " inch Concrete Pipe".

**7** Double-click the value cell of PrtSN.

The Calculation Assistant is displayed.
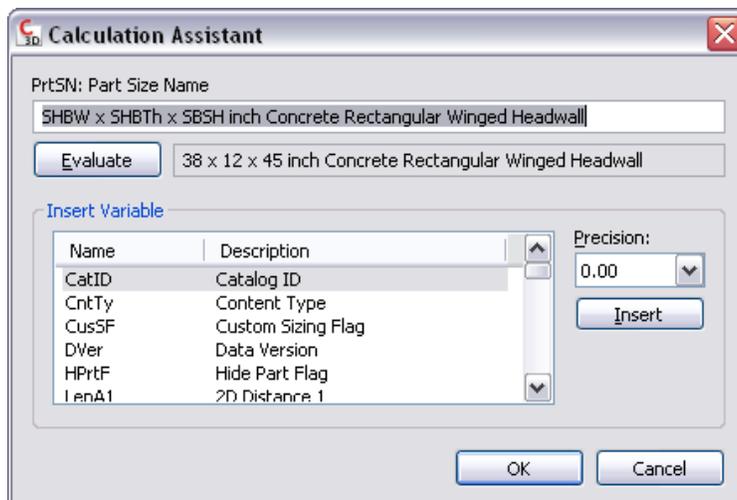
**8** Define the part size name with a calculated value:

- Highlight the value in the PrtSN: Part Size Name text box and press DELETE to remove the text.

- Select 0 for Precision.

- Under Insert Variable, select a variable and click Insert.

- Click the value of PrtSN:Part Size Name and enter the text you want to display in the Part Size Name string. For example, enter inch Dia. Concrete Pipe to add that text to the part size name.

- Under Insert Variable, select Ptype and click Insert.

- Select PTyp and click Insert.

**9** Click Evaluate; the calculation result is displayed.

**10** Click OK.

The calculation value of PrtSN is updated with the valid string and, when selected, the result is displayed in the status bar.
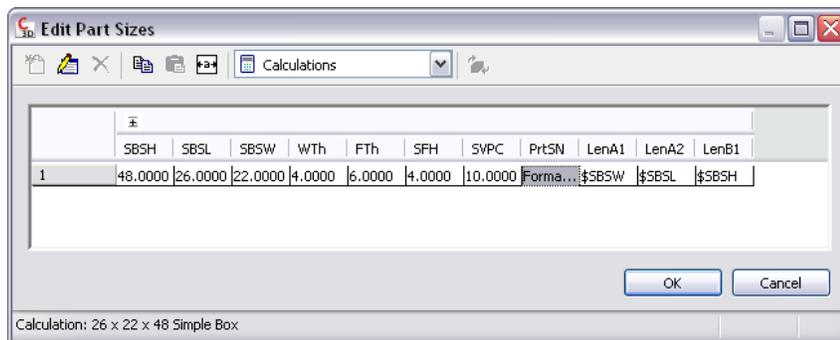


**IMPORTANT** The part size name is generated using VB (Visual Basic) Script's FormatNumber function and simple string substitutions. Correct syntax is crucial. Use the Calculator to ensure that the string is valid.

**11** Click OK.

## Adding a Constraint

Use this procedure to add constraints to the model.

The procedure below uses the example of aligning the top and bottom faces of a rectangular shaped part. For more information on constraints, see Constraints in Understanding Part Builder.

**1** In the part browser, right-click XY Plane and click Add Geometry ➤ Point.

**2** In the modeling area, specify a point near the center of the top face rectangular profile and press Enter.

**3** In the part browser, right-click XY Plane and click Add Constraints ➤ Equal Distance.

**4** In the modeling area, specify the pairs of geometric constraints.

**5** For the first pair, select the point in the center of the top face, and then the lower-left edge of the top face rectangular profile.

**6** For the second pair, select the point in the center of the top face, and then the upper-right edge of the top face rectangular profile.

The model of the top face is updated to the specified constraint.

**7** Repeat steps 3 and 4 to constrain the upper-left and lower-right edges of the top face.

The model of the top face is updated to the specified constraint.

---

**NOTE** When adding constraints, you may be prompted that the geometry is unconstrained by a specific number of dimensions. These prompts are for informational purposes only to assist you during the modeling process. It is recommended that you add a minimum number of constraints to define the shape of the part to avoid undesirable results.

---

**8** To ensure the alignment of the top and bottom faces, in the part browser, right-click Bottom Face and click Add Geometry ➤ Point Reference.

**9** Select the existing point in the center of the top face rectangular profile.

A reference point is added to the bottom face work plane.

**10** Repeat steps 3 through 5 to constrain all four edges of the bottom face using the reference point in the center of the bottom face.

The model of the bottom face is updated to the specified constraints.

## Applying a Modifier

Use this procedure to apply modifiers to the model.

The procedure below uses the example of applying a transition modifier to the two rectangular shaped parts described in Adding a Constraint. For more information on modifiers, see Modifiers in Understanding Part Builder.

**1** In the part browser, right-click Modifiers and click Add Transition.

**2** In the modeling area, select the top face rectangular profile for the start profile; then select the bottom face rectangular profile for the end profile.

A transitional box is created to represent the diffuser, and a transition modifier is added to Modifiers in the part browser.

**3** To add the lip of the diffuser, in the part browser, right-click Work Planes and click Add Work Plane.

The Create Work Plane dialog box is displayed.

**4** Click Offset, enter Lip Offset for Name, and click OK.

**5** In the modeling area, select the XY plane as the reference work plane, drag the cursor above the XY plane, and enter 1 for the offset distance.

The Lip Offset work plane is created.

**6** In the part browser, expand Modifiers, right-click Transition, and click Visible.

The transition modifier display is turned off in the modeling area.

**7** In the part browser, right-click Modifiers, click Add Extrusion, and select the top face rectangular profile.

The Extrusion Modifier dialog box is displayed.

**8** Under Termination, select Plane for Type, select Lip Offset for To, and then click OK.

A box is created to represent the top lip of the diffuser, and an extrusion modifier is added to Modifiers in the part browser.

**9** To see the entire model of the part, in the part browser, right-click Transition and click Visible.

## Generating a Preview Image of a Part

Use this procedure to generate a preview image of the part to help during part selection.

Part Builder generates the preview image based on a specified view direction. You can select from the ten standard AutoCAD view directions to view the model (top, bottom, left, right, front, back, SW isometric, SE isometric, NE isometric, and NW isometric).

1 To create a preview image, on the toolbar, click Generate Bitmap.

The Bitmap Preview dialog box is displayed.

2 Under Generate View, click a view direction for the preview image of the part.
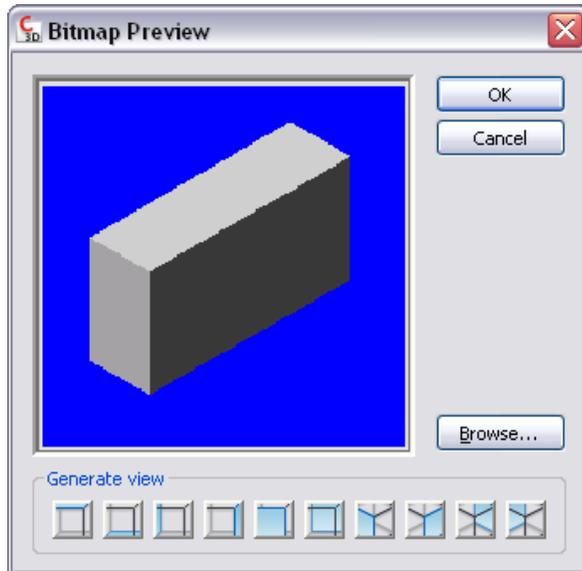


**TIP** As you select a view, the preview image window in the dialog box is updated. This enables you to view all the available preview images for the part before selection.

You can also click Browse to navigate to and select a bitmap image. Predefined images must be 200 x 200 pixels saved with 256 colors.

3 Click OK.

## Defining Part Insertion Position

Use the Part Builder Autolayout commands to define the placement point where a part is inserted into a drawing.

**NOTE** It is helpful to clean up the appearance of the model before defining the placement point. To turn off all work planes and the associated geometry, profiles, and dimensions on those work planes, in the part browser, right-click each feature and click Visible.

1 On the Part Builder toolbar, click Options.

The Options dialog box is displayed.

2 Make sure the check box in the Value column for the Custom Sizing Flag property is unchecked, and that the Hide Part Flag property is checked, then click OK.
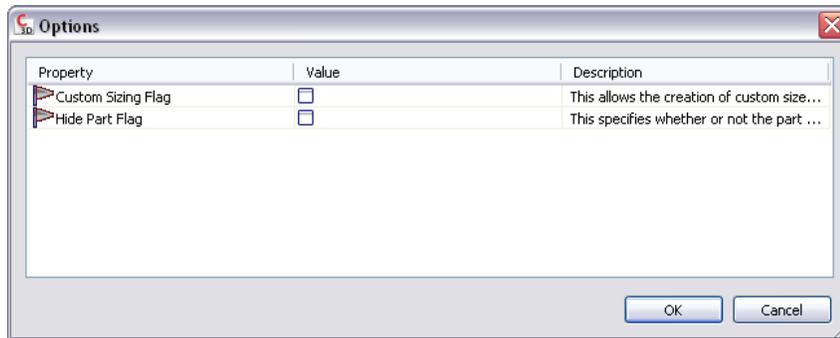
In the part browser, Autolayout Data is added to Modeling, and trim length points are displayed on the model in the modeling area.

**3** Change the model view to plan view. On the View menu, click 3D Views ➤ Plan View ➤ World UCS.

**4** In the part browser, expand Autolayout Data, right-click Layout Data and click Add Trim Length.

---

**NOTE** To ensure components are trimmed correctly when placing a part into a drawing, you must define trim lengths for the part in a specific order—left to right, then bottom to top.

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You are prompted to select the start and end of the trim length. Repeat this for the three trim lengths required for auto layout of the part.

- Define the first trim length.
  For the start of trim length 1, select the point at the center of the part. For the end of trim length 2, select the point at the left end of the part.

- Define the second trim length.
  For the start of trim length 2, select the point at the center of the part. For the end of trim length 2, select the point at the right end of the part.

- Define the third trim length.
  For the start of trim length 3, select the point at the top of the branch (the center of the part). For the end of trim length 3, select the point at the bottom of the branch.

Trim length lines are displayed between the selected points.

**5** In the part browser, right-click Layout Data and click Select Placement Point.

You are prompted to select a point on your model. This point is the location at which connecting segments would intersect if they were extended along their logical paths. The placement point is used as the insertion point for the part when it is added to a drawing during autolayout.

**6** Select the trim length point at the center of the part.

A placement point is displayed at the selected location.

## Validating and Saving a Part

Use this procedure to validate and save the part.

**1** On the Part Builder toolbar, click Validate.

Upon successful completion of these procedures, the part is validated and the status bar is updated with a message indicating if the part validation is successful.

**Part is valid**
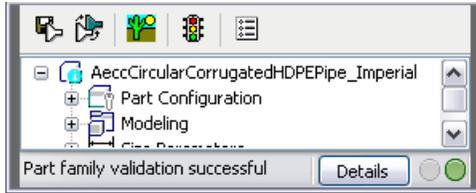


If you have errors in your model, the status bar message displays that the part validation failed, and a dialog box is displayed listing the errors. Review the errors, make necessary modifications, and repeat this step until validation is successful.

**Part is invalid**
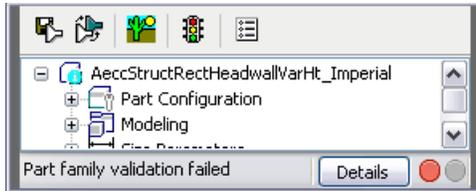


**resulting error dialog box when part is invalid**
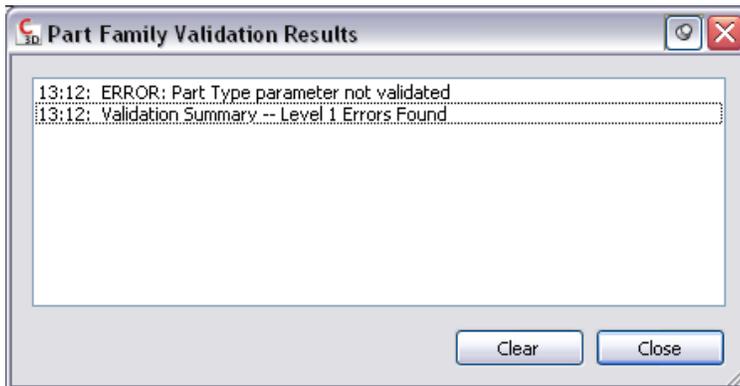


**NOTE** You may have warnings in the Part Family Validation Results dialog box even when the part is valid.

**2** Once validation is successful, on the toolbar, click Save Part Family or Save Part Family As.

**3** Specify whether to make the part available in the catalog.

■ Click Yes to make the part available to users.

■ Click No to keep the part hidden.

**NOTE** This prompt toggles the Hide Part option in the Options dialog box in Part Builder.

The part is saved in the specified catalog location.

**4** Click Close on the File menu.

**NOTE** At this point, you are prompted to save changes to the drawing file <new part family name>.dwg. If you saved the part family, you should also save this drawing file, so click Yes at this prompt. If you did not save the part family, then you should click No at this prompt and not save this drawing file.

# Testing Parts

The guidelines presented here will verify that your part sizes work correctly for your AutoCAD Civil drawings.

It is important to test each part size before using it in your drawings. You should verify the insertion point, and all view representations. If testing produces undesirable results, you should modify and retest the appropriate part sizes.

## Testing the Display Representations

This section summarizes how to test the display representations of parts.

Depending on whether you test part sizes in a drawing started from scratch or from a template, you verify accurate view representations in one of two ways: by scrolling through the various display configurations in a viewport, or by scrolling through the various layout tabs in a template. You should see representations on the Model tab similar to the following as you change display configurations for the viewport.

**TIP** To quickly change display configurations for a viewport, select a display configuration from the list in the bottom-right corner below the drawing area.

## Using Catalog Regen

Catalog Regen is a necessary step that runs through an entire part catalog and validates all parts.

You can run Catalog Regen by clicking the Catalog Regen button on the Getting Started - Catalog Screen dialog box. Use this procedure to regenerate a part catalog that you have modified. Regenerating a part catalog updates the APC file. A part catalog can be used inAutoCAD Civil only if regeneration has been completed.

**To regenerate a part catalog**

1   Verify that the part catalog you modified is the current catalog that is selected in the Part Catalog field of the Getting Started - Catalog Screen dialog box.

    For example, if you modified the pipe catalog, make sure Pipe is selected in the Part Catalog field. If you modified the structure catalog, make sure Structure is selected in the Part Catalog field.

2   On the Getting Started - Catalog Screen dialog box, select the top-level catalog folder in the tree view, and then click the Catalog Regen button. This regenerates the selected part catalog and updates the catalog to reflect new or deleted part sizes.

    The Catalog Regen dialog box is displayed, showing the status of the regeneration process.

3   To verify that the part catalog has been regenerated and updated, open Windows Explorer.

4   Navigate to the part catalog location \Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Pipes Catalog\US Imperial Structures\Validate, and double-click the Validate folder located in the same directory as the appropriate Autodesk part catalog (APC) file.

5   In the Validate folder, open Catalog_Regen_Summary.txt, and scroll through the file to verify that the selected part sizes have been copied to, or deleted from, the part catalog.

## Validating Parts in a Catalog

Use this procedure to validate parts you have changed or added to a part catalog.

Validating a part catalog steps through the catalog structure and verifies its parts. Remember that only valid parts are available during part selection.

**To validate parts in a part catalog**

1   Verify that the appropriate part catalog is the current catalog that is selected in the Part Catalog field of the Getting Started - Catalog Screen dialog box.

    For example, if you want to validate the pipe catalog, make sure Pipe is selected in the Part Catalog field. If you want to validate the structure catalog, make sure Structure is selected in the Part Catalog field.

2   On the Part Builder Getting Started - Catalog Screen dialog box, click the Catalog Test button.

    The Test Catalog feature steps through the catalog structure and validates part sizes by verifying that necessary part size information exists for each part size. The Catalog Test dialog box is displayed, showing the status of the testing process. When the catalog test is complete, an AutoCAD alert dialog box is displayed, reporting the results of the test.

3   To verify that the part sizes in the part catalog have been validated, open Windows Explorer.

4   Navigate to the part catalog location \Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Pipes Catalog\US Imperial Structures\Validate, and double-click the Validate folder located in the same directory as the appropriate Autodesk part catalog (APC) file.

5   In the Validate folder, open Catalog_Validation_Summary.txt, and scroll through the file to verify that all the part sizes have been validated. If a part was not validated, an error notice in the file states which information is missing for the part size.

# Modifying Parts

This section summarizes how to modify existing parts using Part Builder.

For parts in the pipe network catalogs provided by AutoCAD Civil, you can change the part behavior and part size parameters. For parts you create using Part Builder, you can change the part behaviors, part size parameters, and part model including geometry, profiles, modifiers, constraints, and dimensions. You can also delete parts that you no longer need. Deleting parts can be helpful when creating custom catalogs to ensure that all associated definition files are managed correctly.

# Starting Part Builder for Modifying a Part

Use this procedure to start Part Builder and modify a part.

1   Home tab ➤ Create Design panel ➤ ▼ ➤ Part Builder 🔲 .

    The Getting Started - Catalog Screen dialog box is displayed.

2   In the part browser, navigate to and select a part.

3   To modify the part, click 🗗 .

    The existing part is opened in the parametric building environment.

4   To delete a part, click ✖ .

    The selected part family is deleted from the part catalog, including all associated definition files (XML, DWG, and BMP files).

# Making Changes to a Part

As your design develops and parts change, you can modify parts using Part Builder. You can change the part behavior and part size parameters. For parts you created using Part Builder, you can also change the part model, including geometry, profiles, modifiers, constraints, and dimensions. You can use the same procedures that guide you through creating a part to modify it. This section outlines how you can modify a part and provides references to the related procedures that you can use to make changes.

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**IMPORTANT** When making changes to a part, refer to the illustration in Part Modeling (page 1293). This illustration shows the relationship dependencies between features that must be maintained to ensure you are creating a usable part.

---

### Modifying the Part Configuration

You can change the part configuration that controls the behavior of and defines the characteristics of the part. In the part browser, expand Part Configuration and change the part type or subtype. For more information, see Specifying the Part Configuration (page 1309).

### Modifying the Part Model

For parts that you created using Part Builder, you can change the model parameters that define the overall size of the model. You can change the geometry, dimensions, and constraints of the model.

You can add, modify, or remove geometry in the model; however, you must append the changed geometry to the profile in order for Part Builder to update the model and assign new geometric constraints.

You can also change the parametric relationships of model elements by modifying the geometric and dimension constraints. Because constraints control the overall shape of the model, you cannot safely make changes until you know the current constraints applied to the model. You can delete an unwanted constraint or add new constraints to reshape the sketch.

Part Builder's parametric commands ensure that relationships among geometric elements remain intact; however, after changes are made to the model you should re-examine the geometric constraints and dimensions to verify that nothing else in the model needs to be updated.

### Modifying Part Sizes

You can change the model dimensions of the part that determine the overall size of the model, such as length and width. You can also change individual size parameters for a specific part size. For example, you can change a constant size parameter value to a list of values. For more information, see Adding Model Dimensions (page 1311) and Adding Individual Part Sizes (page 1312).

### Modifying the Preview Image or Part Insertion Behaviors

You can change the preview image by selecting a different view direction that Part Builder uses to generate the preview image. For more information, see Generating a Preview Image of a Part (page 1317).

You can change the insertion configuration of the part that defines the placement point of a part in a drawing. For more information, see Defining Part Insertion Position (page 1317).

# Adding a Material Property to a Part

Use this procedure to add the Material property to a part family in the part catalog.

Having the Material property available for parts enables you to use Civl 3D labeling features to add text to identify the type of material used for the pipe. For example, you can configure the Material property to display text such as PVC, ductile iron, or corrugated steel.

Some part families may already have the Material property included by default. However if a part family does not have the Material property included, you can add this property to a part family using the following procedure.

**To add the Material property to a part family**

1 Open Part Builder by selecting Home tab ➤ Create Design panel ➤ ▼ ➤ Part Builder 🔲 .

The Getting Started - Catalog Screen dialog box is displayed.

2 In the Getting Started - Catalog Screen dialog box, open the desired pipe or structure part catalog.

For example, under Pipe ➤ US Imperial Pipe Catalog ➤ Circular Pipes, open the Concrete Pipe catalog by double-clicking it, or by clicking Getting Started - Catalog Screen Modify Part Sizes.

3 In the Part Builder left pane, right-click Size Parameters, and then click Add.

4 On the New Parameter dialog box, click Material, and then click OK.

Notice that when you do this, the Mat (material) property is added to the Edit Part Sizes dialog box. You may need to scroll to the right on this dialog box to see this property added.

5 Now that the Material property is added to this dialog box, you can configure the attributes for this property by clicking in the cell for each attribute.

For example, you may want this Material property to show or not show in AutoCAD Civil drawings. Do this by setting the Visible attribute to True or False.

6 Make sure you set the Visible attribute to True so that the value for the Material property displays in the drawing.

7 Click OK on the Edit Part Sizes dialog box.

8 Click Save Part Family.

Now the Material property is available for parts in this part family. Using AutoCAD Civil labeling features, you can add text to this property to identify the type of material used for the pipe. For more information, see Adding a Material Type to a Pipe or Structure Label (page 1270).

# Corridor Modeling

# Corridors

# 30

You can use AutoCAD Civil corridor modeling to create flexible and configurable 3D models of corridors, such as highways and railways.

Corridor Modeling Workflow

## Understanding Corridor Modeling

A corridor model builds on and uses various AutoCAD Civil objects and data, including subassemblies, assemblies, alignments, surfaces, profiles, and intersections.

A corridor object is created from a baseline (alignment) by placing 2D sections (assemblies) at incremental locations, and by creating matching slopes that reach a surface model at each incremental location:
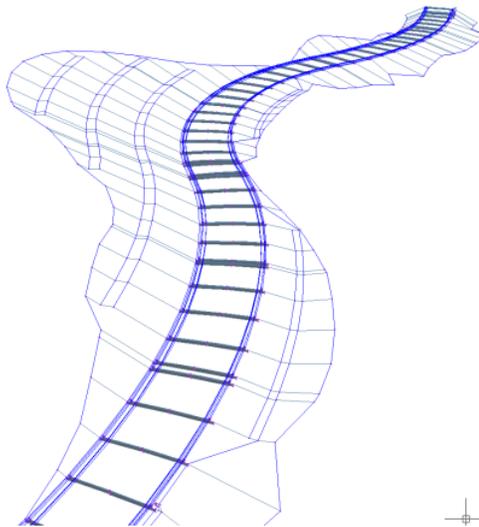


You can also create corridors with multiple baselines, which enables you to create more complex designs, such as intersections (page 1428).
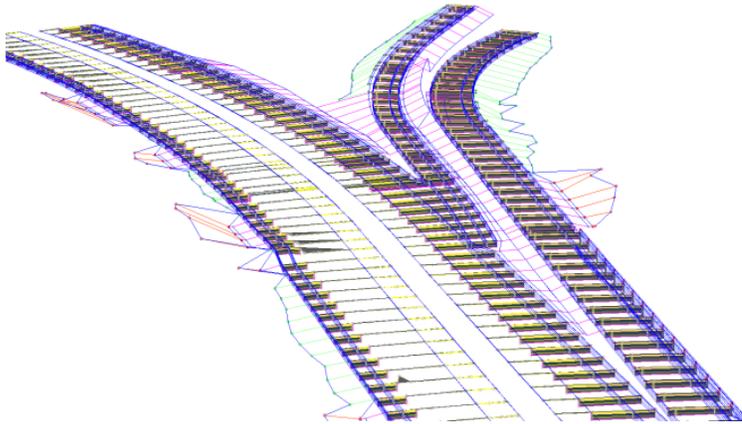
Corridors are created from and based on existing AutoCAD Civil objects, which include:

- **Alignments** (horizontal). Used by a corridor as its centerline. For information, see Alignments (page 855).

- **Profiles** (vertical alignments). Used to define surface elevations along a horizontal alignment. For information, see Creating Profiles (page 1046).

- **Surfaces**. Used to derive alignments and profiles, and for corridor grading. For information, see Surfaces (page 599).

- **Subassemblies**. A fundamental component of a corridor model. Subassemblies define the geometry of a corridor section (assembly). For example, a typical roadway may be composed of paved lanes (on either side of the centerline), a paved shoulder, a gutter and curb, and a roadside grading. These parts are defined independently as subassemblies. You can stack any type of subassembly to make up a typical assembly and apply the same assembly for a station range along an alignment. For more information, see Understanding Subassemblies (page 1394).

- **Assemblies**. Represent a typical section of a corridor. Assemblies comprise one or more subassemblies connected together. For more information, see Understanding Assemblies (page 1377).

After you have created a corridor, you can extract data from it, including surfaces, feature lines (as polylines, alignments, profiles, and grading feature lines), and volume (quantity takeoff) data. For information, see Exporting Corridor Data (page 1369) and Analyzing Sectional Volumes (page 1167).

Corridors have their own display style and also inherit styles from their components. Before creating corridors, you should be familiar with creating and managing styles (page 53) and command settings (page 97).

## The Corridor Object

Corridors persist in an AutoCAD drawing as objects with the name AeccCorridor.

Corridor objects are defined by associating a baseline (alignment) with sectional design elements, and other structural data. The corridor object manages the data, tying various assemblies (applied for different ranges of stations) to the baselines (alignments) and their finished grade profiles. It manages the connection of project-specific surface and alignment data to the subassembly and assembly content. The object includes corridor body geometry, longitudinal feature lines, embedded surfaces, rendering support, and slope hatching support.

A corridor can define and display components, such as:

■ Feature lines connecting points along the point codes, which are defined in the subassemblies (used to create the assemblies).

■ Surfaces, using link codes and feature lines.

For overview information about AutoCAD Civil objects, see Understanding Objects and Styles (page 51).

## Corridor Styles and Display

How corridor elements are displayed is controlled both by a specific corridor style and the styles and labels associated with the objects used to create the corridor.

Use corridor styles to control the appearance of corridor region boundaries, assembly insertion stations, and stations where the default properties of the subassemblies are overridden.

For more information, see Corridor Style Dialog Box (page 1824).

In addition, various components within the corridor are controlled by their own display styles and labels:

■ Display styles for corridor components include alignments, code sets, slope patterns, surfaces, and feature line styles.

■ Components of the corridor object have their own labels. Corridor surface objects support all annotation capabilities that a regular surface supports.

The following table lists the styles of the corridor components and where they are derived from.

| Corridor Component | Display Style | Notes |
| --- | --- | --- |
| Links | Link style | Set through associated Code Set style. |
| Points | Marker style | Set through associated Code Set style. |
| Shapes | Shape style | Set through associated Code Set style. |
| Feature Line | Feature Line style | Defined in the corridor properties. |
| Surfaces | Surface style | Defined in the corridor properties and in the surface properties, if a dynamic surface is generated. |
| Material Rendering | Render Material | Defined in the corridor properties. |
| Corridor Section | Code Set style; assigned from corridor. | View of an assembly, so all elements, such as links, points, and shapes, are displayed through respective styles in the set. |
| Surface Section | Section style | |

For more information about styles, see Object Styles (page 53).

## Corridors Collection (Prospector Tab)

Use the Corridors collection in the Prospector tree to access the corridors in a drawing or a project. As corridor objects are created, they are displayed in the Corridors collection.

Expand the Corridors collection to view the names of the corridors and display a tabular list of the corridors in the Prospector list view. For more information, see The Toolspace Item View (page 82).

## Corridor Collection (Settings Tab)

Use the Corridor collection in the Settings tree to manage corridor settings, styles, and command settings.

Right-click the Corridor collection to do the following:

■ Edit the corridor feature settings

■ Refresh the display of the settings tree.

For more information about the contents of this shortcut menu, see "The Object Collection (Settings Tree)".

Expand the Corridor collection to display and edit the styles and command settings that are available for corridors.

| For information about... | See... |
| --- | --- |
| Corridor settings | Editing Corridor Settings (page 1333) |
| Corridor styles | Corridor Styles and Display (page 1329) |

# Creating Corridors

You can use two methods to create a corridor: using the Create Corridor command or the Create Simple Corridor command.

All corridors are managed in the same way, regardless of how you create them. Also, all corridors are listed in the Corridors collection in the prospector tree.

Before you can create a corridor, you must create the underlying data, such as surfaces, alignments, profiles, subassemblies, and assemblies.

| For information about creating... | See... |
| --- | --- |
| Surfaces | Creating Surfaces (page 601) |
| Alignments | Creating Alignments (page 887) |
| Profiles | Creating Layout Profiles (page 1048) |
| Subassemblies | Creating Subassemblies (page 1396) |
| Assemblies | Creating Assemblies (page 1380) |

For information about how a corridor is displayed after it is created, see Changing Corridor-Related Default Style Settings (page 1335).

# Creating a Simple Corridor

Use the Create Simple Corridor command to quickly create a basic corridor.

This method is ideal for creating basic corridors based on a single alignment (1), a profile (2) along the alignment, and an assembly (3) across it. You can subsequently add more complexity to the corridor by editing it.
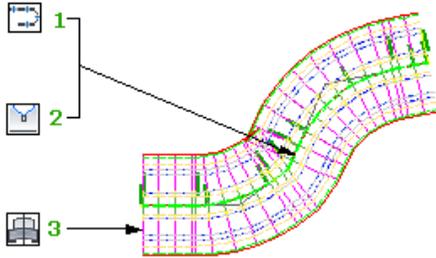


For information about adding more complexity, see Managing and Editing Corridors (page 1333).

**NOTE** Before you create a corridor, you must create the underlying data, such as surfaces, alignments, profiles, and assemblies.

**To create a simple corridor**

1 Click Home tab ➤ Create Design panel ➤ Corridor drop-down ➤ Create Simple Corridor 🔧 .

2 In the Create Simple Corridor Dialog Box (page 1830), in the Name field, enter a name for the corridor.

  To name the corridor, select its default name and enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

3 In the Description field, enter a description for the corridor.

4 To change the style used by the corridor, click the Corridor Style list or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

  The corridor style controls the appearance of corridor region boundaries and assembly insertion stations. For more information, see Corridor Styles and Display (page 1329).

5 Click 📚 to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

  **NOTE** If you do not select a layer, the corridor is placed on the default layer.

6 Click OK.

7 Select an alignment in the drawing or press Enter to select an alignment in the Select An Alignment dialog box (page 1810).

8 Select a profile in the drawing or press Enter to select a profile in the Select A Profile dialog box (page 1838).

9 Select an assembly in the drawing or press Enter to select an assembly in the Select An Object dialog box (page 1810).

10 The corridor name is displayed in the Corridors collection on the Prospector tab. For information about editing and managing the corridor, see Managing and Editing Corridors (page 1333).

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Corridor drop-down ➤ Create Simple Corridor 

Menu

Corridors menu ➤ Create Simple Corridor

Command Line

CreateSimpleCorridor

Dialog Box

Create Simple Corridor (page 1830)

## Creating a Corridor

Use the Create Corridor command to specify complex parameters at creation time.

You specify the station range, profile, assembly, and any necessary targets.
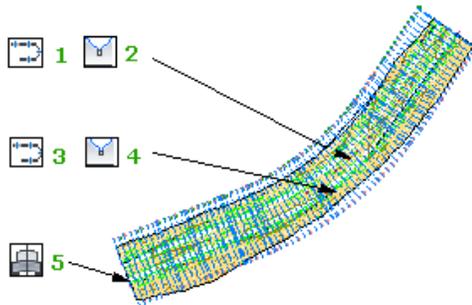


Other parameters could include station frequency and controlling offsets, as well as information about multiple baselines and regions.

**To create a corridor**

1 Click Home tab ➤ Create Design panel ➤ Corridor drop-down ➤ Create Corridor  .

2 Select an alignment in the drawing or press Enter to select an alignment in the Select An Alignment dialog box (page 1810).

3 Select a profile in the drawing or press Enter to select a profile in the Select A Profile dialog box (page 1838).

4 Select an assembly in the drawing or press Enter to select an assembly in the Select An Assembly dialog box (page 1810).

5 In the Create Corridor dialog box (page 1826), in the Name field, enter a name for the corridor.

To name the corridor, select its default name and enter a new name, or use the name template. For more information, see Name Template Dialog Box (page 1806).

6 In the Description field, enter a description for the corridor.

7 Click  to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

NOTE If you do not select a layer, the corridor is placed on the default layer.

**8** To change the style used by the corridor, click the Corridor Style list or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

The corridor style controls the appearance of corridor region boundaries and assembly insertion stations. For more information, see Corridor Styles and Display (page 1329).

**9** Edit the parameters in the properties table. You can add additional controlling baselines, regions, or offsets, modify alignments, profiles, assemblies, or station ranges and frequency, or edit targets. For more information, see Editing Corridor Parameters (page 1336).

**NOTE** If targets are required for the corridor and not set, messages are displayed in the Event Viewer. To display the Event Viewer, click General ➤ Utilities ➤ Event Viewer. For more information about the event viewer, see The Event Viewer Vista (page 1626).

**10** Click OK to create the corridor.

The corridor name is displayed in the Corridors collection on the Prospector tab.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Corridor drop-down ➤ Create Corridor

Menu

Corridors menu ➤ Create Corridor

Command Line

CreateCorridor

Dialog Box

Create Corridor (page 1826)

# Managing and Editing Corridors

You can control the default settings and styles for creating corridors and also edit or add data to individual corridors after you create them.

Use the Corridor Properties dialog box to set:

■ a corridor's administrative information

■ parameters such as baselines, frequencies, and targets

■ code sets

■ corridor feature lines, surfaces, boundaries, and slope patterns

# Editing Corridor Settings

Use corridor settings to specify the default behavior for corridor-related commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Toolspace Settings tree. You can control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

In the Settings tree, use the Corridor collection's shortcut menu to establish defaults for all corridor-related commands. You can change corridor-specific settings at this level, or override the drawing ambient settings.

Use the Commands collection in the Corridor collection to change corridor settings for a specific command.

**NOTE** Overrides to the drawing ambient settings at the Corridor collection level and the Commands collection level affect only the specified (and subordinate) levels. The drawing level settings are not changed.

The topics in this section describe only those settings that affect corridor-related commands. They do not apply to the drawing ambient settings that you can change at the Corridor collection level. For more information about the drawing ambient settings, see Specifying Ambient Settings (page 68).

## Changing Corridor Creation Settings

Use the corridor creation and naming settings in the Edit Feature Settings - Corridor dialog box to change corridor-related settings before you create corridors.

For example, before creating a corridor, you may want to specify the default assembly frequency along tangents, curves, or spirals.

If a closed lock appears in the Lock column for a property, the property is locked at a higher level of the Settings tree. It cannot be changed at this level.

If you change a property value, a check mark is placed in the Override column. This override is also noted in the property table for related objects higher up the Settings tree, where an arrow is displayed in the Child Override column.

**To change corridor creation settings**

1  Do one of the following:
   ■  **To edit settings for all corridor-related commands**: In Toolspace, on the Settings tab, right-click the Corridor collection. Click Edit Feature Settings.

   ■  **To edit settings for a specific command**: In Toolspace, on the Settings tab, expand the Corridor collection. Expand the Commands collection. Right-click the name of the command and click Edit Command Settings. For example, right-click the CreateCorridor command and click Edit Command Settings.

   An Edit Feature Settings - Corridor dialog box (page 1830) is displayed.

2  To specify the default highlight settings for corridor region baselines, boundaries, targets and internal assemblies; right-click the Corridor collection and click Edit Feature Settings. Edit the Region Highlight Graphics settings.

3  To specify the default assembly frequency along tangents, curves, and spirals, right-click the CreateCorridor or CreateSimpleCorridor commands and click Edit Command Settings. Edit the Assembly Insertion Defaults settings.

4  To specify whether assemblies are to be inserted at specific points, such as horizontal geometry or superelevation critical points, right-click the CreateCorridor or CreateSimpleCorridor commands and click Edit Command Settings. Edit the Assembly Insertion Defaults settings.

5  To edit the default scale to view corridor sections, right-click the ViewEditCorridorSection command and click Edit Command Settings. Expand the View/Edit Options property group. Edit the Default View Scale setting.

6  To specify whether a corridor is automatically rebuilt when you edit a subassembly parameter, right-click the ViewEditCorridorSection command. Click Edit Command Settings. Expand the View/Edit Options property group. Edit the Rebuild On Edit setting.

**7** To specify the front and back clip values for 3D objects displayed in a corridor section view, right-click the ViewEditCorridorSection command and click Edit Command Settings. Expand the View/Edit Options property group. Edit the Front Clip and Back Clip settings.

**8** To specify whether the station tracker is on, right-click the ViewEditCorridorSection command. Click Edit Command Settings. Expand the View/Edit Options property group. Set the Turn Off Unassociated Layers setting to no and the Station Tracker in Multiple Viewports setting to yes.

**9** To specify the grid and grid text display settings for a corridor section view, right-click the ViewEditCorridorSection command and click Edit Command Settings. Expand the Grid Settings and Grid Text Settings property groups. Edit the grid line, grid spacing, color, and text size settings.

**10** To edit the default naming format for corridors and data that can be extracted from the corridor, expand the Default Name Format property group. Edit the Corridor Name Template settings.

**11** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Corridor collection ➤ Edit Feature Settings
OR
Settings tab: Corridor ➤ Commands ➤ Right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Corridor (page 1830)

## Changing Corridor-Related Default Style Settings

Use the corridor settings to specify default styles and labels for corridor creation and for data that you create or extract from corridors.

You can select the default styles for alignments and profiles exported from corridors and a default corridor style and label set.

The default styles specified in these settings are used to establish the default setting for the style in the Create Corridor and Corridor Properties dialog boxes.

**To change the corridor-related style and label settings**

**1** In Toolspace, on the Settings tab, right-click the Corridor collection. Click Edit Feature Settings to open the Edit Feature Settings - Corridor dialog box (page 1830).

**2** Expand the Default Styles property group.

**3** To specify a default style for corridor output objects, such as alignments and profiles generated from corridors, click the corresponding field. Click ⬚ to open the Select Style dialog box.

**4** To specify a default corridor style, click Corridor Style. Click ⬚ to open the Select Style dialog box.

**5** To specify a default style for a label set, click the corresponding field. Click ⬚ to open the Select Label Set dialog box.

**6** To specify a default style for viewing corridor sections, click Section View Style. Click ⬚ to open the Select Style dialog box.

**7** Click Apply.

### Quick Reference

## Editing Basic Corridor Information

After creating a corridor, use the Prospector tree to edit a corridor's name and description.

**To edit basic corridor information**

**1** In Toolspace, on the Prospector tab, expand the ![icon] Corridors collection. Right-click the corridor. Click Properties.

**2** In the Corridor Properties dialog box, click the Information tab (page 1815).

**3** To change the name of the corridor, enter a new name in the Name field.

**4** To change the description of the corridor, enter a new description in the Description field.

**5** To change the style of the corridor, click the Object Style list. Alternately, or use the standard buttons to create, edit, or select a style.

**6** Click Apply to make the changes, or click OK to make the changes and close the Corridor Properties dialog box.

### Quick Reference

## Editing Corridor Parameters

Edit the parameters of the current corridor, including the selected baselines, regions, and controlling offsets.

A corridor is defined by at least one baseline (alignment) and an assembly that is applied for a range of stations on that baseline. In many cases, corridors will have different assemblies at different stations, depending on the existing ground and other design considerations. Also, it may be necessary to build a corridor model that is controlled by multiple baselines, for example, one that includes intersection objects (page 1424). To add and edit this type of complexity, use the Parameters tab, where you can modify the associated alignments, profiles, and assemblies, change assembly frequency and range, and update targets.

# Adding and Editing Corridor Regions

Add additional regions to a corridor or modify the parameters (such as start, end, and station frequency, assembly association, and targets) of existing regions.

Corridor regions are used to associate assemblies to specific station ranges along the corridor. Therefore, to assign different assemblies to sections of the corridor, you create regions.

To edit corridor regions, use the commands:

### Edit Region

Edit the parameters of the selected region including; alignment, profile, assembly, start and end stations, the frequency to apply assemblies, target mapping, and station overrides.

### Isolate Region

This command turns off the visibility of all corridor regions, except the selected one. This command has the same effect as clearing all region check boxes on the Parameters tab of the Corridor Properties dialog box, except that of the selected region.

### Show All Regions

This command restores the visibility of any regions whose visibility is currently turned off. The command is not available if all regions are visible.

**To add a corridor region**

1   In the drawing, click the corridor object, right-click, and click Corridor Properties.

2   In the Corridor Properties dialog box, click the Parameters tab (page 1815). Do one of the following:
   ■   To add a new region after any existing regions, right-click the baseline for which you want to add a new region. Click Add Region.

   ■   To insert a new region between two existing regions, right-click the first region. Click Insert Region.

3   In the Select An Assembly dialog box, select the assembly to assign to the region.

   The region is added to the corridor.

   ---
   **NOTE** If the assembly has offsets defined, the offsets are displayed as children of the region. For information about creating an assembly offset, seeCreating an Assembly Offset (page 1382). For information about editing a corridor region's offset, see Editing Offsets (page 1341)
   ---

4   Edit the region. For information, see To Edit a Corridor Region (page 1337).

**To edit a corridor region**

1   In the drawing, select the corridor object you want to edit.

2   Click Corridor tab ➤ Modify panel ➤ Edit Region .

3   In the Corridor Properties dialog box, click the Parameters tab (page 1815).

4   To edit a selected region from the drawing, click  Select Region From Drawing and select the region in the drawing.

5   To change the assembly for the region, click the Assembly field. Select the assembly in the Select An Assembly dialog box.

**6** To change either the start or end station for the region, select either the Start Station or End Station field. Enter the station number or click ⌖ and click the station in the drawing.

**7** To change the station frequency, click Set All Frequencies. Specify the frequency in the Frequency To Apply Assemblies dialog box (page 1836).

**8** To change the target to which the region is mapped, click Set All Targets. Specify the targets in the Target Mapping dialog box (page 1840).

If you do not change the target when you change an assembly, the target will remain the same for most common target types.

**9** To delete a region, select the region, right-click, and click Remove Region.

**10** Click Apply to make the changes.

**To isolate a corridor region to edit it**

**1** In the drawing, select the corridor region you want to edit.

**2** Click Corridor tab ➤ Modify panel ➤ Isolate Region ⬚ .

**3** On the Modify panel, click Edit Region ✐ .

**4** In the Corridor Properties dialog box, Parameters tab (page 1815), edit the region. Click OK to make the changes.

**5** Click Corridor tab ➤ Modify panel ➤ Show All Regions ⬚ .

All corridor regions are again visible.

> **NOTE** You may have to rebuild (page 1361) the corridor to restore the visibility of all its regions.

**To add stations to a region**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In the Corridor Properties dialog box, click the Parameters tab (page 1815).

**3** Click the Frequency field for the region to which you want to add stations.

**4** In the Frequency To Apply Assemblies dialog box (page 1836), click ⊹ . Select the station location in the drawing.

**5** To add a description for the station, click the Description field. Enter the description.

**6** Click OK.

**To split a region**

**1** In the drawing, select the corridor object you want to split.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Properties drop-down ➤ Corridor Properties ⬚ .

**3** In the Corridor Properties dialog box, click the Parameters tab (page 1815).

**4** In the table, right-click a corridor region. Click Split Region.

**5** In the drawing, click a point or points parallel to the alignment to split the selected region.

**6** Right-click to stop selecting split point(s).

**7** Click OK to split the region.

## Quick Reference

Ribbon

Corridor tab ➤ Modify panel ➤ Corridor Properties drop-down ➤ Corridor Properties

Corridor tab ➤ Modify panel ➤ Edit Region

Corridor tab ➤ Modify panel ➤ Isolate Region

Corridor tab ➤ Modify panel ➤ Show All Regions

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

Corridor Properties

Command Line

EditCorridorProperties
EditCorridorRegion
IsolateCorridorRegion
ShowAllCorridorRegions

Dialog Box

Corridor Properties - Parameters tab (page 1815)

## Editing Corridor Regions Using Grips

When you select a corridor region, triangular grips are displayed at the start and end stations of the region. Use these grips to manually change the start and end station for a region.

If the start station of one region is the end point of another region, the grip at the station is displayed as a diamond shape, which enables you to edit the shared extents of the adjacent regions at the same time.
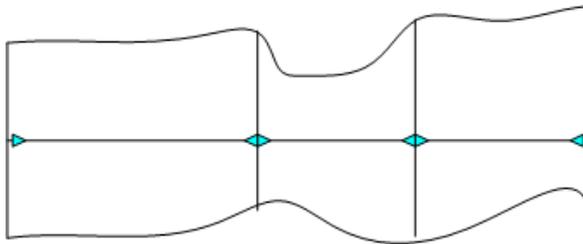


If you want to edit the start or end point of a region without editing the adjacent region, additional grips enable you to edit the extents of the regions independently.
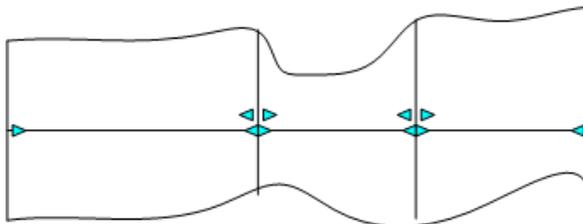
**To edit a corridor region using editing grips**

**1** In the drawing, select the corridor object.

Grips are displayed at the start and end stations of the region.

**2** Ctrl+click a grip to make it active, and drag it to a new location.

**3** If the grip is diamond-shaped, the start of one region and the end of another are both at this location. You can edit the two regions at the same time, or edit the extents of the regions independently.

## Adding Baselines

Add additional baselines to a corridor to create offset alignments.

Typically, you need to add additional baselines if you require offsets or more complex corridor models, such as ones with intersections.

**To add baselines**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In the Corridor Properties dialog box, click the Parameters tab (page 1815).

**3** To add a new baseline, click Add Baseline.

**4** In the Pick Horizontal Alignment dialog box, select the alignment from the list or click  to select an alignment in the drawing.

**5** To select the profile for the baseline, click the Profile field. Select the profile from the list or click  to select a profile in the drawing.

**6** To add a region to the baseline, right-click the baseline. Click Add. For information, see Adding and Editing Corridor Regions (page 1337).

**7** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Parameters tab (page 1815)

## Hiding and Showing Regions

Hide or show corridor regions or entire baselines.

If a region or baseline is hidden, it is not updated if underlying elements, for example, alignments and profiles, are modified.

---

**NOTE** The check boxes you use to hide a region have a tri-state display. If a baseline has multiple regions and only one of the regions is selected, the check box is dimmed and not available.

---

**To hide corridor regions or baselines**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In the Corridor Properties dialog box, click the Parameters tab (page 1815).

3 To hide a region or baseline, clear the check box next to the region or baseline name.

---

**NOTE** The hide/display check boxes have a tri-state display. If there are multiple regions under a baseline and only one of the regions is selected, the check box is dimmed.

---

4 Click Apply.

5 To display a hidden region or baseline, select the check box next to the region or baseline.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Parameters tab (page 1815)

## Editing Offsets

If a region has an assembly offset specified, you can edit the alignment, profile, and station start and end for the offset.

For information, on assembly offsets, seeCreating an Assembly Offset (page 1382).

**To edit an offset alignment for a region**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In the Corridor Properties dialog box, click the Parameters tab (page 1815).

3 To change the alignment for the offset, click the Alignment field. Select the alignment in the Pick Horizontal Alignment dialog box.

4 To change either the start or end station for the region, select either the Start Station or End Station field. Enter the station number or click ⬚ and select the station in the drawing.

5 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Parameters tab (page 1815)

## Changing Station Frequency

Add or modify the frequency of stations in a corridor region.

Stations frequency can be specified by referencing the existing alignment and profile geometry:

- **Along tangents**. Specify the assembly insertion frequency along the tangent portion of an alignment.
- **Along curves**. Specify the assembly insertion frequency along the curve portion of an alignment.
- **Along spirals**. Specify the assembly insertion frequency along the spiral portion of an alignment.
- **Along profile curves**. Specify the assembly insertion frequency along the curve portion of the profile.
- **At specific points**. Specify whether assemblies should be inserted at specific geometry points, including horizontal geometry, superelevation critical, profile geometry, and profile high and low points.

---

**NOTE** If both an assembly frequency setting that references horizontal geometry and a setting that references vertical geometry apply to a portion of a corridor, the setting that results in a smaller interval is used. For example, if a segment is both a horizontal tangent and a vertical curve, and the vertical curve frequency adds assemblies at more frequent intervals, the frequency specified for vertical curves is used.

---

You can also modify station location and frequency by manually adding stations.

**To change the station frequency for a region**

1   In the drawing, click the corridor object, right-click, and click Corridor Properties.

2   In the Corridor Properties dialog box, click the Parameters tab (page 1815).

3   Click the Frequency field for the region to which you want to add stations.

4   In the Frequency To Apply Assemblies dialog box (page 1836), modify the properties as required.

5   To manually add a station, click ⊕ . Select the station location in the drawing.

6   To add a description for the station, click the Description field. Enter the description.

7   Click OK.

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

  Corridor Properties

Dialog Box

  Corridor Properties - Parameters tab (page 1815)

## Viewing and Deleting Overridden Stations

View and delete stations that have assembly overrides applied.

Assembly overrides are created and edited using the View/Edit Corridor Section Tools. However, you can view and delete overrides in the Corridor Properties Parameters tab.

For information about creating assembly overrides, see Editing Corridor Sections (page 1365).

**To view or delete assembly overrides for a region**

1   In the drawing, select the corridor object, right-click, and click Corridor Properties.

2   In the Corridor Properties dialog box, click the Parameters tab.

3   For a region that has assembly overrides, click the Overrides column. For information about creating assembly overrides, see Editing Corridor Sections (page 1365).

4   In the Assembly Override Stations dialog box (page 1814), the assembly name, range, stations, and override method are displayed.

5   To remove the assembly overrides from a station (and apply the default assembly parameters) and delete the station from the list, click the station and click ✖ .

6   To remove all the assembly overrides from the region, click Delete All.

The list is cleared and the default assembly parameters are applied to all stations in the corridor region.

7   Click OK to close the dialog box.

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

  Corridor Properties

Dialog Box

  Assemblies Overrides (page 1814)

## Setting and Editing Targets

On a corridor, you can set and edit the values of subassembly target objects, such as surfaces, alignments, and profiles.

Targets are required when the geometry of one or more of an assembly's subassemblies requires corresponding surface, offset, or elevation targets for defining that geometry. A few subassemblies even use pipe networks as target objects. When a corridor contains subassemblies that use targets, the object names of the intended target objects must be mapped from the subassembly definition to the corresponding drawing objects. This task is referred to as setting targets or mapping targets.

**NOTE** If targets required for the corridor are not set, messages are displayed in the Event Viewer, indicating that target objects are not found. To display the Event Viewer, click View tab ➤ Palettes panel ➤ ▼ ➤ Event Viewer ⟁ . For more information about the event viewer, see The Event Viewer Vista (page 1626).

For example, a subassembly 'SideSlope' parameter is used to define roadside grading, starting from the edge of shoulder, and sloping at a specified grade, until it intercepts with an existing ground surface. In the subassembly, references to the target surface are made by calling a surface object named 'TargetDTM.' However, when you create the corridor, and apply the subassembly to a corridor definition, there may not be a surface named 'TargetDTM', and there may be multiple ground surfaces. Therefore, after the assembly is applied to the corridor, you need to map the target name (TargetDTM) to an actual surface object name.

Typically, subassembly parameters that can use a target object to define a width or an offset can use the following types of objects to define that width or offset: alignments, polylines, feature lines, or survey figures.

Similarly, subassembly parameters that can use a target object to define an elevation can use the following types of objects to define that elevation: profiles, 3D polylines, feature lines, or survey figures.

Subassemblies that can use a target object to define a surface can only use a surface object to define that surface.

As previously mentioned, a few subassemblies allow you to use pipe network objects as targets, such as the TrenchPipe subassemblies.

The following table summarizes these capabilities:

| Target Type | Object types that can be used as targets |
| --- | --- |
| widths or offsets | alignments, polylines, feature lines, or survey figures |
| elevations | profiles, 3D polylines, feature lines, or survey figures |
| surfaces | surfaces |
| pipe networks | pipe networks |

The Help topic for each subassembly lists the parameters in the subassembly that can be associated with a target object.

**To set or edit targets**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In the Corridor Properties dialog box, click the Parameters tab (page 1815)

3 Do one of the following:
- To view or map all targets required by the corridor, click Set All Targets.
- To view or map the targets for a specific baseline or assembly, click the Target field for that baseline or assembly.

4 In the Target Mapping dialog box (page 1840), set the surface, offset, and elevation targets in the corresponding Object Name fields.

5 Click OK to close the Target Mapping dialog box.

6 Click Apply.

**To select a surface to target**

1 In the Target Mapping dialog box (page 1840), do one of the following:
- To select surfaces individually, click the Object Name field next to the corresponding surface in the Target column.

■ To select all surfaces at once, click <Click Here To Set All> in the Object Name field. This is useful for mapping TargetDTM's that are referenced by multiple subassemblies used in the corridor through multiple assembly regions. For example, all side slope subassemblies typically need to be mapped to the same existing ground surface.

**2** In the Pick a Surface dialog box, select the surface from the list or click  and select the surface in the drawing.

**3** Click OK to close the Pick a Surface dialog box.

**To select a width or offset to target**

**1** In the Target Mapping dialog box (page 1840), click the Object Name field next to the corresponding width or offset target name in the Target column.

**2** To target an alignment:

■ In the Select Object Type To Target list of the Set Width Or Offset Target dialog box (page 1840), select Alignment.

■ Select an alignment from the Alignment list, or click  and select an alignment in the drawing, to add to the list.

■ Click Add >> to add an alignment to the Selected Entities to Target list.

**3** To target a feature line, survey figure or polyline:

■ In the Select Object Type To Target of the Set Width Or Offset Target dialog box select, Feature Lines, Survey Figures and Polylines.

■ Click Select By Layer to import all named targets on any layer selected in the Select By Layer, dialog box or click Select From Drawing to select entities in the drawing.

■ Click  to remove any excess entities from the Selected Entities To Target list.

■ If there are two or more targets in the Selected Entities To Target list, select Target To Nearest Object or Target To Farthest Object for Selection Choice If Multiple Targets Are Found.

**4** Click OK to close the Set Width Or Offset Target dialog box.

**NOTE** If you select more than one target object, **Varies** is displayed in the Object Name field.

**To select a slope or elevation to target**

**1** In the Target Mapping (page 1840) dialog box, click the Object Name field next to the corresponding slope or elevation target name in the Target column.

**2** To target a profile:

■ In the Set Slope Or Elevation Target dialog box (page 1839), in the Select Object Type To Target list, select Profiles.

■ Select from the Select An Alignment list, or click  and select an alignment in the drawing.

■ Select from the Select Profiles list, or click  and in the drawing select a profile to add to the list.

■ Click Add >> to add a profile to the Selected Entities To Target list.

**3** To target a feature line, survey figure or polyline:

- In the Select Object Type To Target of the Set Slope Or Elevation Target dialog box select, Feature Lines, Survey Figures and Polylines.

- Click Select By Layer to import all named targets on any layer selected in the Select By Layer dialog box or click Select From Drawing to select entities in the drawing.

- Click to remove any excess entities from the ✖ Selected Entities to Target list.

- If there are two or more targets in the Selected Entities To Target list, select Target To Nearest Object or Target To Farthest Object for Selection Choice If Multiple Targets Are Found.

**4** Click OK to close the Set Slope or Elevation Target dialog box.

**NOTE** If you select more than one target object, **Varies** is displayed in the Object Name field.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Parameters tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Parameters tab (page 1815)

## Editing the Code Set Style

View and modify the code set style for the corridor.

A corridor is defined using a set of subassemblies. These subassemblies can have styles assigned to points, links, and shapes. You may want to display these components in a variety of ways. Code set styles enable you to manage the display these components by using a mapping system that maps code to style.

For information about code set styles and subassemblies, see Using Codes and Code Set Styles (page 1405).

**To edit a corridor's code set style**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In the Corridor Properties dialog box, click the Codes tab (page 1817). All codes used by the corridor with their corresponding descriptions, style, and label style information are displayed in the properties table.

**3** To change the code set style used by the corridor, click the Code Set Style list or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

These styles control the way the corridor is displayed in section view.

**4** Click ◫ to view information for the selected style.

**5** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Codes tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Codes tab (page 1817)

## Editing Feature Lines

Use the Feature Lines tab to view the feature line codes and display styles or to modify how a corridor's feature lines are connected.

By default, AutoCAD Civil joins all points with the same point code along a baseline and displays them as feature lines. You can modify how points are connected when a point code is used multiple times at consecutive stations.

### Editing Feature Line Connections

Modify feature line connections by specifying which point codes are connected using branching options and the direction for branching, and whether extra point codes are to be connected.

**Branching and Connecting Extra Points**

If an assembly is applied at consecutive stations with a varying number of point codes (of the same type), you can control both whether the feature lines are connected and how they are connected.

The following illustrations show different branching and point connection options and the resultant feature lines.

In all illustrations:

- Assemblies with a varying number point codes of the same type are used at the stations.

- The centerline (heavy black line) is the corridor baseline (alignment).

- The circular points on the gray lines are of one type of point code.

- The circular points on the green lines are of another type point code.

- There are two groups in each assembly: left and right (of the baseline).

- The Connect option is selected for both types of point codes.

Inward branching and extra points (with the same point code) are not connected:
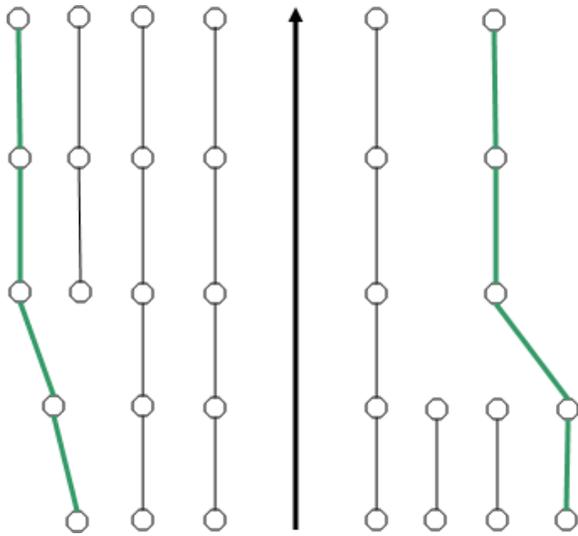
Inward branching and extra points (with the same point code) are connected:
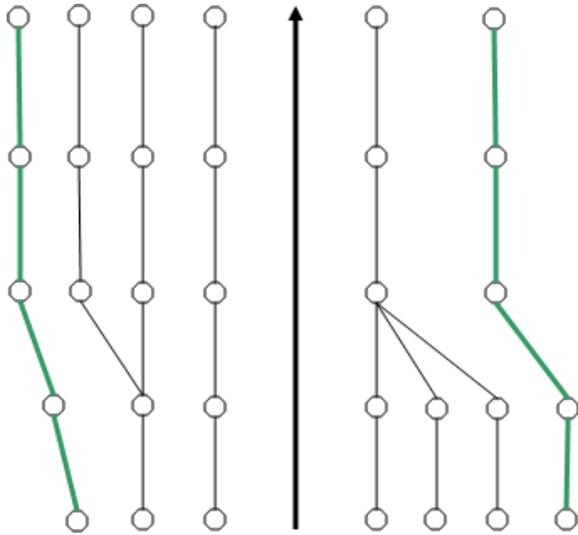
Outward branching and extra points (with the same point code) are not connected:
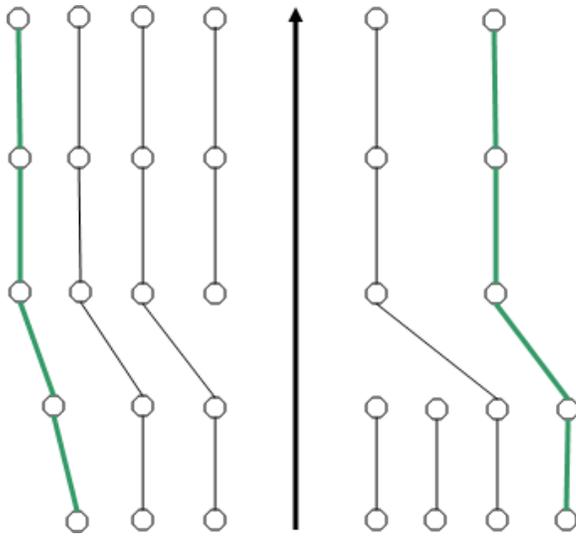
Outward branching and extra points (of the same point type) are connected:
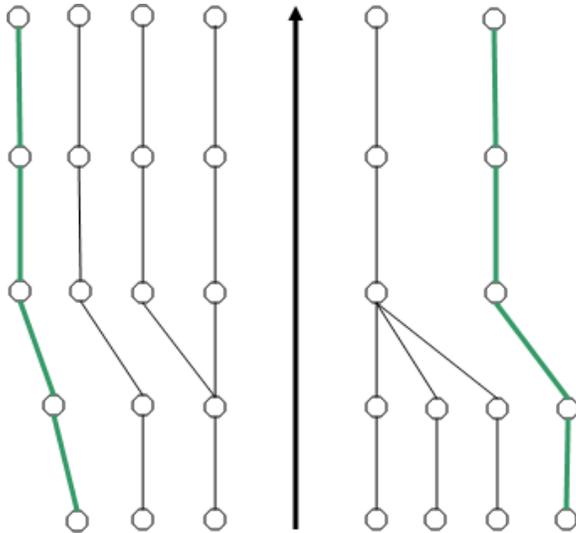


**To edit the feature line connection**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In the Corridor Properties dialog box, click the Feature Lines tab (page 1818).

3 By default, all identical point codes are connected as longitudinal feature lines. To delete feature lines between one or more types of point codes, clear the check box in the Connect field for the appropriate code.

---

**NOTE** If the Connect check box is disabled, the feature line is being used in a corridor surface definition. It cannot be disconnected.

---

4 To control how to connect point codes that are used a varying number of times at different sections, use the Branching field. Select one of the following:

■ **Inward**: The feature line branches inward, joining the innermost points.

■ **Outward**: The feature line branches outward, joining the outermost points.

**5** To prevent feature lines from being disconnected when there is branching, select Connect Extra Points.

**6** Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Feature Lines tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Feature Lines tab (page 1818)

## Editing Feature Line Styles

View and modify the feature line styles for the corridor.

Use different feature line styles to distinguish the feature lines for easier selection and identification. The feature line style for each code is controlled by the code set style.

**To edit the feature line styles**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In Corridor Properties dialog box, click the Feature Lines tab (page 1818). All point codes with their corresponding types and feature line style information are displayed in the grid.

**3** To edit feature line styles applied to a point code, click the Code Set Style list, or use the standard controls to create or edit a style.

**4** To change the style for an individual code, for the point code you want to change, click  in the Feature Line Style field. Select a style in the Pick Feature Line Style dialog box. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

**5** Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Feature Lines tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Feature Lines tab (page 1818)

## Creating and Editing Corridor Surfaces

Use the Surfaces tab of the Corridor Properties dialog box to create corridor surfaces.

Because corridor surfaces are an output of a corridor model, they remain dynamically linked to the corridor object. Any changes to the corridor definition are reflected in the surface definitions. You can also create a detached surface from a corridor surface.

For information about creating a detached surface, see Exporting Corridor Surfaces (page 1373).

When you create a corridor surface, it is added to the Surfaces collection on the Toolspace Prospector tab. You can work with a corridor surface the same way you do with any surface in the Surfaces collection, including changing its style, adding labels to it, and using it for surface analysis. The following features and behaviors are unique to corridor surfaces:

■ When you select a corridor surface, only the surface is selected. The corridor it is based on is not selected.

■ When you change the surface style of a corridor surface using its surface properties, the style is also changed on the corridor properties Surfaces tab.

■ When a corridor is rebuilt, corridor surfaces are updated to reflect any changes in the corridor, and then any edits are applied to the corridor model.

■ The corridor from which the surface was taken is listed in the surface properties definition.

---

**NOTE** Do not use a corridor surface as a target for assemblies.

---

## Creating a Corridor Surface

You can create a corridor surface and then add the required feature line or link codes. Also, you can create a separate corridor surface from each link code in a single operation.

■ **Create a corridor surface**. Use this method if a corridor surface is to be extracted from a set of link codes and corridor feature lines. For example, create a surface using the Links data type and the Top code. After a surface is created, you can add more link codes and feature lines to the surface definition.

■ **Create a corridor surface from each link code**. Use this method to create one surface based on each link code. For example, if Top, Paved, and Side_slope are defined as link codes (in the subassemblies used to build the corridor), this method creates three surfaces: one surface using Top as breaklines, a second surface using Paved, and a third surface using Side_slope.

**To create a corridor surface**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In Corridor Properties dialog box, click the Surfaces tab (page 1818).

3 Optionally, click  to open the Name Template dialog box. Set or modify the corridor surface naming convention. For information about the name template, see Name Template Dialog Box (page 1806).

4 Click  to create an empty corridor surface.

5 To add data to the surface, select it in the grid. Select the data type from the Data Type list. Select the code from the Specify Code list. Click  .

6 In the grid, expand the surface to view its data components.

7 Optionally, to add additional data to the corridor surface, repeat Step 5.

8 Optionally, to create another surface and add data to it, repeat Steps 4 and 5.

9 If you do not want to create a dynamic corridor surface, clear the check box next to the surface name.

10 Click Apply to display the corridor surface in the drawing.

If the check box beside the surface name is selected, the surface is added to the Surfaces collection on the Toolspace Prospector tab.

**To create a corridor surface for each link**

1   In the drawing, click the corridor object, right-click, and click Corridor Properties.

2   In Corridor Properties dialog box, click the Surfaces tab (page 1818).

3   Optionally, click to open the Name Template dialog box. Set or modify the corridor surface naming convention. For information about the name template, see Name Template Dialog Box (page 1806).

4   Click to create a corridor surface from each link code. Corridor surfaces are created from all link codes and are listed in the grid. You can view and modify them.

5   If you do not want a corridor surface added to the Surfaces collection on the Toolspace Prospector tab, clear the check box next to that surface name.

6   Click Apply to display the corridor surfaces in the drawing.

   If the check box next to a surface name is selected, the surface is added to Surfaces collection on the Toolspace Prospector tab.

## Quick Reference

Toolspace Shortcut Menu

   Prospector tab: Corridors  ➤  <corridor name>  ➤  Properties  ➤  Surfaces tab

Object Shortcut Menu

   Corridor Properties

Dialog Box

   Corridor Properties - Surfaces tab (page 1818)

## Copying Corridor Surfaces

Create a copy of an existing corridor surface

**NOTE** Because a corridor has a large number of dependencies, it cannot be moved, copied, rotated, nor scaled up or down. However, a corridor surface can be copied. You can then edit the copied surface.

When a corridor surface is copied, a naming convention is used. For example if the original corridor surface is named 'Corridor - Datum', the new surface is named 'Corridor - Datum (1)'. If the 'Corridor - Datum' surface is copied again, the new surface is named 'Corridor - Datum (2)'. If the 'Corridor - Datum (1)' surface is copied, the new surface is named 'Corridor - Datum (1) (2)'.

**To copy a corridor surface**

1   In the drawing area, click the corridor surface. Right-click and click Basic Modify Tools  ➤  Copy.

   At the command line, you are prompted to specify a base point for the corridor surface.

2   Click an insertion point in the drawing area.

   The corridor surface is pasted in the drawing and added as a new surface to the Surfaces collection in the Prospector tree.

**3** To edit the name of the corridor surface, right-click the surface in the Prospector tree. Click Properties.

The Surface Properties dialog box is displayed with the Information tab (page 2346) active. The Name field contains the name of the copied corridor, which is, by default, " <surface-name> (1)."

**4** Enter a new name in the Name field.

**5** To specify the description of the corridor surface, enter a new description in the Description field.

**6** Click Apply to make the changes, or click OK to make the changes and close the Surface Properties dialog box.

## Editing a Corridor Surface

Modify corridor surfaces to change how they are displayed and to add or remove data from the surfaces.

When you create a corridor surface as a dynamic surface, you can edit many of the surface properties and perform surface analysis using the surface properties. Access surface properties using the Surfaces collection in the Toolspace Prospector tab. For more information, see Understanding Surfaces (page 599)

**To edit a corridor surface**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In Corridor Properties dialog box, click the Surfaces tab (page 1818).

**3** In the properties table, select the surface that you want to edit.

**4** To modify the name of the surface, click the name. Modify accordingly.

**5** To change the surface display style used by the corridor surface, click the Surface Style field. Select the style from the Pick Corridor Surface Style dialog box or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

**6** To change the render material used by the corridor surface, click the Render Material field. Select the style from the Select Render Material dialog box or use the standard style creation tools to edit or create a style. For more information about rendering corridor surfaces, see the Rendering Corridor Models (page 1374).

**7** To add data to the surface, select the surface in the properties table. Select the data type in the Data Type list. Select the code in the Specify Code list. Click ⊕ .

**8** To remove data from a surface, in the properties table, expand the surface. Select the data component that you want to delete, and click ✖ .

**9** To remove the corridor surface from the Surfaces collection on the Toolspace Prospector tab, clear the check box next to the surface name.

**10** Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Surfaces tab

Object Shortcut Menu

Corridor Properties

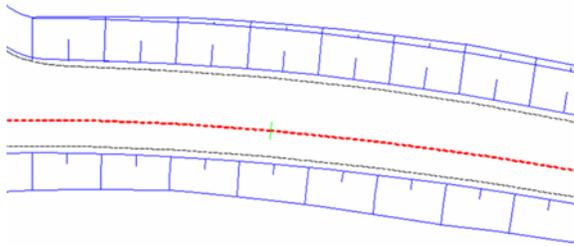Dialog Box

Corridor Properties - Surfaces tab (page 1818)

Command Line

DefineCorridorSurfaces

## Resolving an Overhanging Surface

Correct overhanging surfaces in the corridor.

When subassemblies are joined to form an assembly, surfaces of the subassemblies can overhang one another. This presents an incorrect representation of how the surface should be modeled.

Overhanging surfaces are corrected in the Surfaces tab, Corridor Properties dialog box. You can correct the overhang by following the top links or bottom links in the assembly. The overhang correction feature defaults to no links.
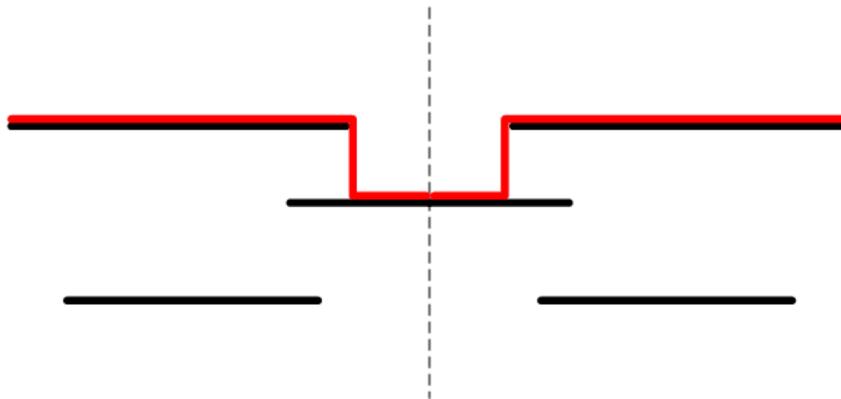
**Surface (red) corrected to top links**
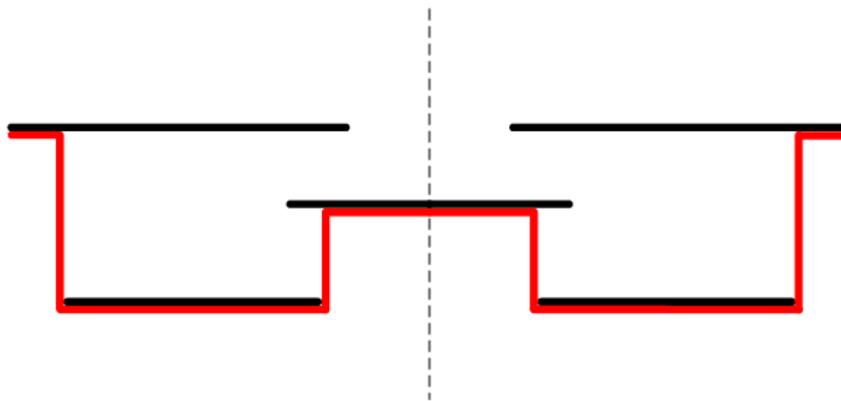
**Surface corrected to bottom links**

**To resolve an overhanging surface**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In Corridor Properties dialog box, click the Surfaces tab (page 1818).

3 In the Name column, select the corridor surface.

4 Click in the Overhang Correction column, and select Top Links or Bottom Links. The entry defaults to None.

**5** Click OK to correct the overhang.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Surfaces tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Surfaces tab (page 1818)

## Adding and Editing Corridor Boundaries

Use corridor surface boundaries to prevent triangulation outside of the daylight lines of a corridor surface and prevent them from being displayed or to render an area of the corridor surface as a render material.

Corridor surfaces support the following types of boundaries:

■ **Render Only**. Used to represent different parts of corridor surface with different materials (when rendering), for example, asphalt and grass. For more information about rendering, see Rendering Objects (page 1579).

■ **Hide Boundary**. Used to mask to create void areas or punch holes in the corridor surface. For example, if a link code Paved is used on either side of the corridor with another surface (a median), separating them, when you create a corridor surface using Paved as the data, AutoCAD Civil tries to connect the gap in between two link codes. To create voids, you define boundaries to represent the surface appropriately.

■ **Outside Boundary**. Used to define the outer boundary of the corridor surface.

> **NOTE** A corridor surface must exist before you can add a corridor boundary. For information about creating a corridor surface, see Creating and Editing Corridor Surfaces (page 1350).

## Adding Boundaries to a Corridor Surface

Use the Boundaries tab of the Corridor Properties dialog box, or the Create Boundary From Corridor command on the Create Design panel of the Home tab, to create corridor boundaries.

In the Boundaries tab, you can create the boundaries:

■ Automatically, when there is a single baseline, based on point codes to which feature lines are assigned

■ Corridor Extents As Outer Boundary, when there are multiple baselines, based on feature lines or surface link codes

■ Interactively, by selecting feature lines in the drawing

■ From Polygon, using a closed polygon that forms the boundary

When selecting the feature lines from which to create the boundary, first select the bottom end of one feature line, followed by the top end of its pair.

For example, select the bottom outside edge of a lane followed by the top outside edge of a lane to create a boundary that encompasses both lanes:
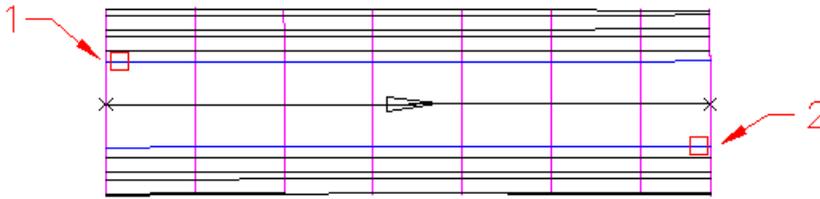
If you are creating the corridor boundary using a polygon, you can create the required polygon by exporting corridor feature lines as polylines, and then join them to create the required polygon.

For more information about converting feature lines to polylines, see Exporting Corridor Data (page 1369).

**NOTE** The option to add boundaries Automatically is not available if more than one baseline is used in the corridor. In that case, select Corridor Extents As Outer Boundary, to automatically add a boundary for each corridor surface.

**To automatically create outer boundaries**

1   In the drawing, click the corridor object, right-click, and click Corridor Properties.

2   In the Corridor Properties dialog box, click the Boundaries tab (page 1820).

3   Right-click the corridor surface to which you want to add a mask.

■   To automatically add a boundary to a corridor with a single baseline, click Add Automatically ➤ <code name>.

**NOTE** Only point codes that form a pair of feature lines in the corridor are listed, for example daylight and ETW.

■   To automatically add a boundary to a corridor with multiple baselines, click Corridor Extents As Outer Boundary.

The boundary is added to the corridor surface where you can modify its name, style usage and type. For information, see Editing a Corridor Boundary (page 1357).

**To interactively add a boundary to a corridor surface**

1   In the drawing, click the corridor object, right-click, and click Corridor Properties.

2   In Corridor Properties dialog box, click the Boundaries tab (page 1820).

3   Right-click the corridor surface to which you want to add a boundary. Click Add Interactively.

**NOTE** A corridor surface must exist before you can add a corridor boundary. For information about creating a corridor surface, see Creating and Editing Corridor Surfaces (page 1350).

4   In the drawing, click a corridor feature line that will form one side of the boundary. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

5   In the drawing, click the corridor feature line that will form the other side of the boundary. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

6   When you finish selecting the feature lines, press Enter.

7   In the Boundaries tab grid, expand the corridor surface to display the boundary.

8   Click Use Type. Select the type of boundary, either: Render Only, Hide Boundary, or Outside Boundary.

**9** Click Apply.

**To create outer boundaries using a closed polygon**

**1** In the drawing, click the corridor object, right-click, and click Corridor Properties.

**2** In the Corridor Properties dialog box, click the Boundaries tab (page 1820).

**3** Right-click the corridor surface to which you want to add a mask. Click Add From Polygon.

**4** In the drawing window, click the closed polygon you want to use to define the boundaries of the corridor surface.

**5** The boundary is added to the corridor surface where you can modify its name, render style, and type. For more information, see Editing a Corridor Boundary (page 1357).

---

**NOTE** If you create an invalid boundary, you will be prompted to continue (and correct the error), to save the invalid boundary (which can be corrected later) or to cancel the boundary creation.

---

**To create a polyline boundary from a corridor**

**1** Click Home tab ➤ Create Design panel ➤ 🔻 ➤ Create Boundary From Corridor.

**2** Click a corridor in the drawing, or press Enter to select a corridor in the Select A Corridor dialog box.

A closed polyline is created as the outer boundary of the corridor. This closed polyline can then be used to create a surface boundary, using the Add From Polygon (page 1357) command.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ 🔻 ➤ Create Boundary From Corridor

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Boundaries tab

Object Shortcut Menu

Corridor Properties

Command Line

CreateBoundaryFromCorridor

Dialog Box

Corridor Properties - Boundaries tab (page 1820)

## Editing a Corridor Boundary

Rename, change styles, view, and modify the boundary definition.

Because boundaries are connected based on how the feature lines are selected, you can use the Corridor Boundary Definition dialog box to iteratively modify, preview, and validate the boundaries after you create them. You can:

■ Add or remove feature lines from the boundary definition

■ Specify if the boundary is to be drawn on a feature line in reverse of the station incrementing order.

- Change the draw order of feature lines in the boundary.

- Preview the boundary. The boundary is temporarily drawn as a green polygon.

- Ensure that the boundary is well-formed and has no crossing edges.

**To edit a corridor boundary**

1 In the drawing, click the corridor object, right-click, and click Corridor Properties.

2 In the Corridor Properties dialog box, click the Boundaries tab (page 1820).

3 Expand the corridor surface for which you want to modify a boundary.

4 To modify the boundary name or description, click its Name or Description field. Enter a new name or description.

5 To change the render material, which is used for Render Only boundaries, click the Render Material field for the boundary. Select a render material from the Select Render Material dialog box.

6 To view the boundary definitions and change the extents, click **[...]** in the Definition column.

7 In the Corridor Boundary Definition dialog box (page 1813), to change either the start or end points for the boundary, either select the Start Point or End Point fields and enter the values number, or click and click the point in the drawing.

8 To reverse the boundary draw direction along a feature line, either select or clear the Reverse Direction check box.

9 To add a new feature line to the boundary definition, click . Click the feature line in the drawing.

10 To delete a feature line from the boundary definition, select the feature line. Click .

11 To move a feature line up or down in the boundary definition order, select the feature line. Click to move it up or to move it down.

12 To preview the corridor boundary, click .
The boundary is displayed as a green polyline.

13 To verify that the boundary polygon is well formed (for example, that its edges do not cross each other), click .
A message is displayed next to the button, indicating the state of the boundary.

14 Click OK to close the Corridor Boundary Definition dialog box.

15 To change the boundary type, click Use Type. Select the boundary type.

16 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Boundaries tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Boundaries tab (page 1820)

## Creating and Editing Corridor Slope Patterns

Add slope patterns between a set of feature lines.

Slope patterns are slope indicator lines. They have one or more repeating lines that are aligned with the flow direction. The lines can be the length of the slope or less. They can have a predefined symbol or an AutoCAD block inserted at one end. Slope patterns are applied between any two feature lines (typically, between a grading footprint and daylight line). For more information about slope patterns, see Creating Grading Styles (page 737).

Typical slope pattern usage along a corridor:



## Adding a Corridor Slope Pattern

Add slope patterns to corridor surfaces.

**To add a corridor slope pattern**

1  In Toolspace, on the Prospector tab, expand  Corridors, right-click the corridor, and click Properties.

2  In Corridor Properties dialog box, click the Slope Patterns tab (page 1820).

3  Click Add Slope Pattern >>.

4  In the drawing, select the first corridor feature line that will form one side of the slope. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

5  Select the second corridor feature line that will form the other side of the slope. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

6  Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Slope Patterns tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Slope Patterns tab (page 1820)

## Editing a Corridor Slope Pattern

Modify the display and extents of a slope pattern.

**To edit a corridor slope pattern**

1   In Toolspace, on the Prospector tab, expand ![icon] Corridors. Right-click the corridor. Click Properties.

2   In the Corridor Properties dialog box, click the Slope Patterns tab (page 1820).

3   To change the slope pattern style, click ![icon] in the Slope Pattern Style field. Select a style from the Pick Style dialog box or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

4   To change either the start or end points for the slope pattern, select either the Station Start or Station End fields. Enter the values, or click ![icon] and click the location in the drawing.

5   Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ <corridor name> ➤ Properties ➤ Slope Patterns tab

Object Shortcut Menu

Corridor Properties

Dialog Box

Corridor Properties - Slope Patterns tab (page 1820)

## Rebuilding a Corridor

When you make changes to a corridor, such as editing data or the build parameters, the corridor may become out-of-date. You can rebuild it.

If the corridor definition is out-of-date, a warning is displayed next to the corridor's name in the Prospector tree. The corridor needs to be rebuilt.

In AutoCAD Civil, you can rebuild a corridor either automatically or manually.

**To automatically rebuild a corridor**

1   In Toolspace, on the Prospector tab, expand the Corridors collection.

2   Right-click the corridor with the warning ![icon] .

3   Click Rebuild - Automatic.

> **NOTE** If a check mark is displayed next to the Rebuild - Automatic menu item, it is enabled. Then, when changes are made to the corridor or to one or more of its baselines, profiles, or dependant surfaces, the corridor is automatically rebuilt and updated.

**To manually rebuild a corridor**

**1** In Toolspace, on the Prospector tab, expand the Corridors collection.

**2** Right-click the corridor with the warning ⚠ .

**3** Click Rebuild.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ right-click <corridor name> ➤ Rebuild

Object Shortcut Menu

Rebuild Corridor

Command Line

UpdateCorridor

# Deleting a Corridor

You can delete a corridor, erasing it from the drawing and removing it from the Corridors collection in the Prospector tree.

**To delete a corridor**

■ In Toolspace, on the Prospector tab, expand the Corridors collection. Right-click the corridor. Click Delete.
The corridor is erased from the drawing and removed from the Corridors collection in the Prospector tree.

**NOTE** You cannot delete objects that have dependencies (other objects). For example, if you have created a section object from the corridor, you must delete the section before you delete the corridor.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Corridors ➤ right-click <corridor name> ➤ Delete

# Viewing and Editing Corridor Sections

You can use the Corridor Section Editor to view and modify corridor sections.

Use the Corridor Section Editor to apply assembly overrides to a corridor section or a range of sections and then view only those sections. To view both the corridor section view and its location along the corridor (1), open two horizontal viewports.
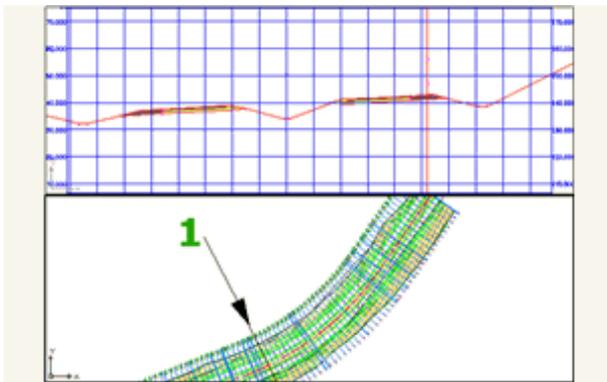
## Viewing Corridor Sections

Use the Corridor Section Editor to visually inspect how assemblies are applied at various stations.

A simple corridor section as viewed with the Corridor Section Editor.
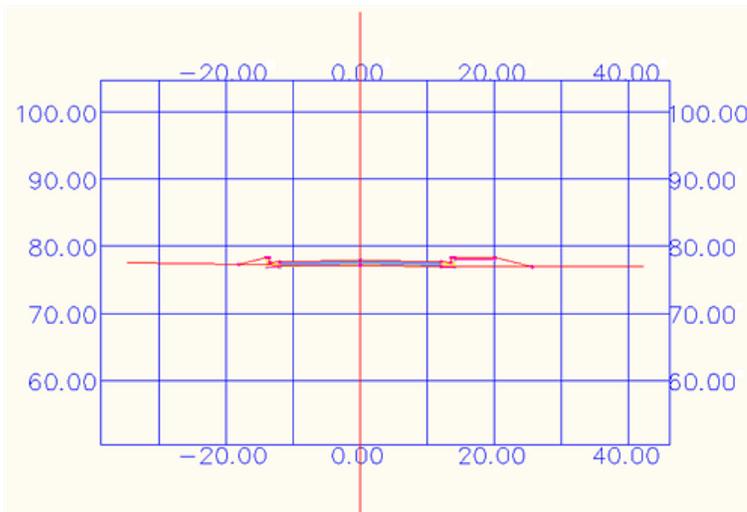


You can step through corridor stations or jump to any station of interest.

**To view a corridor by sections**

1   Select a corridor in the drawing.

2   Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor .

3   Optionally, you can use the Select A Baseline list to select another baseline for the corridor or click and select an alternate alignment in the drawing.

Alternately, in the prospector or the viewport, you can right click on the corridor you wish to view as sections, right-click and click View/Edit Corridor Section.

**To change your view of corridor sections**

1   To view a section at a different station: On the Station Selection panel, select it from the Select a Station drop-down list.

**2** On the Station Selection panel, click ▯◁ or ▷▯ to view the first or end station.

**3** Click ◁ to view the previous station or click ▷ to view the next station.

**4** To view only the stations that have assembly overrides applied to them, click ▯ Show Overridden Stations. Only stations with assembly overrides are displayed in the list.

**5** To view all stations, click ▯ Show Overridden Stations again.

**6** To set zoom options, on the View Tools tab, either:

■ Click ▯ Zoom to Extents to view the entire corridor section view.

■ Click ▯ Zoom to an Offset and Elevation to lock the zoom to a selected elevation or offset.

■ Click ▯ Zoom to Subassembly to lock the zoom to a selected subassembly.

**7** Optionally, to view detailed information about the assembly and subassemblies used by the region in the Corridor Parameters vista: On the Corridor Edit Tools panel, click ▯ Parameter Editor.

**To edit corridor section viewing and editing options**

**1** To change corridor section viewing and editing options: On the Section Editor tab, View Tools panel, click ▯ View/Edit Options.

**2** In the View/Edit Corridor Section Options (page 1841) dialog box, under View/Edit Options, select a style from the Code Set Style list, to change the display of corridor section elements such as shapes, links, and points.

**3** Under View/Edit Options, enter a new scale in the Default View Scale field.

**4** Under View/Edit Options, select the Rebuild on Edit option, to automatically rebuild the corridor model when you edit a subassembly using the section viewing and editing tools.

> **NOTE** To change the default section viewing scale or automatic corridor rebuild value, on the Toolspace Settings tab, expand Corridor ➤ Commands. Right-click ViewEditCorridorSection, and click Edit Command Settings. In the Edit Command Settings dialog box, expand the View/Edit Options property group. Modify the Default View Scale and Rebuild On Edit settings as required.

**5** Under View/Edit Options, edit the values in the Front Clip and Back Clip fields, to display 3D objects, such as pipe network parts located at the specified station.

You can display any 3D object located at the station, including pipe network parts, blocks, and polylines. The clip values control the start and end of the visible extents of objects in the corridor section, from front to back.

> **NOTE** Unless 3D Objects reside on a visible layer, they are not displayed in the corridor section view. By default, layers for objects that are not referenced by the corridor are not visible.

**6** Under Grid Settings, set the visibility and separation of grid lines, the color of grid lines, and the visibility and color of a center axis for the grid.

**7** Under Grid Text Settings, set the style, color and size of text, along with the visibility of center axis annotation.

### Quick Reference

Ribbon

Corridor tab ➤ Modify panel ➤ Corridor Section Editor 📷

Menu

Corridors menu ➤ View/Edit Corridor Section

Command Line

ViewEditCorridorSection

Dialog Box

Corridor Section Editor Ribbon (page 1822)
View/Edit Corridor Section Options (page 1841)

## Tracking Corridor Sections at Stations

Use the Station Tracker and Corridor Section Editor to track the location of corridor sections in plan and profile view.

Station Tracker displays (in both plan and profile view) the location of the station from which the current corridor section view is sampled. Station Tracker can be used only with multiple viewports, where one viewport contains the corridor section view and one or two others contain plan and profile views.

**To track corridor section views in plan and profile views**

1   Change the ViewEditCorridorSection command settings:

   ■   In the Corridor settings, expand the Commands folder.

   ■   Right-click ViewEditCorridorSection and click Edit Command Settings.

   ■   In the Edit Command Settings - ViewEditCorridorSection dialog box, expand the View/Edit Options property group.

   ■   Set the Turn Off Unassociated Layers setting to No.

   ■   Set the Station Tracker in Multiple Viewports setting to Yes.

   ■   Click OK.

2   Set the view to multiple viewports:

   ■   Click View tab ➤ Viewports panel ➤ New 📷 .

   ■   In the Viewports dialog box, New Viewports tab, Standard Viewports list, select Three: Above.

   ■   Click OK.

3   Set the corridor plan view in one of the smaller viewports, and the profile view in the other smaller viewport.

4   Select a corridor in the top viewport.

5   Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor 📷 .

6   In the Section Editor tab, Station Selection panel, Select a Station list, select a station.

   That station's corridor section view will be displayed in the top viewport. Station Tracker displays the station location in both plan and profile views.

**Quick Reference**

Ribbon

Corridor tab ➤ Modify panel ➤ Corridor Section Editor 

View tab ➤ Viewports panel ➤ New 

Menu

Corridors menu ➤ View/Edit Corridor Section
View menu ➤ Viewports ➤ Named Viewports

Command Line

ViewEditCorridorSection
+vports 0

Dialog Box

Corridor Section Editor Ribbon (page 1822)

## Editing Corridor Sections

You can use the Corridor Section Editor to override corridor and assembly parameters and apply the overrides to a station or range of stations.

Edit corridor and assembly parameters as required, using the following methods:

■ Edit the assembly numerically by editing the subassembly parameters.

■ Edit the assembly manually by adding links and points to the subassembly part geometry.

■ Resize assembly parameters by grip editing the subassembly part geometry.

■ Insert or remove subassemblies from a corridor section.

Overriding assemblies may require you to edit the way points, links, and shapes are represented in the corridor as feature codes. Use the Edit User Defined Codes command to change or delete codes as required.

**NOTE** If you have edited a subassembly by either adding or deleting a point, adding or deleting a link, or editing the subassembly geometry using grips, you cannot edit the values of the changed parameters in the Corridor Section Editor.

**To override subassembly parameters by editing their values**

1 Select a corridor in the drawing.

2 Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor  .

3 To view detailed information about the assembly and subassemblies used by the region in the Corridor

Parameters vista: On the Section Editor tab, Corridor Edit Tools panel, click  Parameter Editor.

4 Click the station drop-down list and select the station for which you initially want to override one or more assembly parameters.

5 In the Corridor Parameters vista, click the Value field for the parameter that you want to override. Enter a new value. When an override is applied, the Override column displays True. To remove an individual override, clear the check box next to the subassembly or parameter. To remove all overrides in an assembly, clear the checkbox in the Override column next to the assembly name.

**6** To apply the override to a range of stations: On the Section Editor tab, Corridor Edit Tools panel, click ![icon] Apply to a Station Range.

**7** In the Apply To A Range Of Stations dialog box, enter both the start station and the end station. Click OK.

**8** To view only the stations that have assembly overrides applied to them: On the Station Selection panel, click ![icon] Show Overridden Stations. Only stations with assembly overrides are displayed in the list.

**9** To view all stations, click ![icon] Show Overridden Stations again.

**10** If the corridor is not set to rebuild automatically, click ![icon] Update Corridor to rebuild the corridor.

Corridor surfaces and any other data that is derived from the corridor are updated using the overrides.

**To override subassembly parameters by adding or deleting points**

**1** Select a corridor in the drawing.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor ![icon] .

**3** To view detailed information about the assembly and subassemblies used by the region in the Corridor Parameters vista: On the Section Editor tab, Corridor Edit Tools panel, click ![icon] Parameter Editor.

**4** In the Section Editor tab, Station Selection panel, Select a Station list, select the station to which you want to add a point.

**5** To add a point: On the Section Editor tab, Corridor Edit Tools dropdown, click ![icon] Add Point.

**6** In the drawing window, select the link you want to break by adding a point.

**7** Enter a point code value for the new point.

The new vertex is added at the midpoint of the link. The points at the end of the links retain the codes from the original link.

In the Corridor Parameters vista, the Override column displays ![icon] True.

**8** To delete a point: On the Corridor Edit Tools dropdown click ![icon] Delete Point.

**9** In the drawing window, click the point you want to delete.

The point and all links connected to it are deleted.

**10** To view only the stations that have assembly overrides applied to them: On the Station Selection panel, click ![icon] Show Overridden Stations. Only stations with assembly overrides are displayed in the list.

**11** To view all stations, click ![icon] Show Overridden Stations again.

**12** If the corridor is not set to rebuild automatically, on the Corridor Edit Tools panel, click ![icon] Update Corridor to rebuild the corridor. Corridor surfaces and any other data that is derived from the corridor are updated using the overrides.

**To override subassembly parameters by adding or deleting links**

**NOTE** You can add links only to individual subassemblies. You cannot add links that connect two subassemblies.

**1** Select a corridor in the drawing.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor .

**3** In the Section Editor tab, Station Selection panel, Select a Station list, select the station to which you want to add a link.

**4** To add a link: On the Corridor Edit Tools dropdown, click  Add Link.

**5** In the drawing window, select the first point of the two points you want to connect.

**6** Select the second point.

The link is added.

**7** To delete a link: On the Corridor Edit Tools dropdown, click  Delete Link.

**8** In the drawing window, click the link you want to delete.

The link is deleted. The points that were connected to create the link are not deleted.

**9** To view only the stations that have assembly overrides applied to them: On the Station Selection panel, click  Show Overridden Stations. Only stations with assembly overrides are displayed in the list.

**10** To view all stations, click  Show Overridden Stations again.

**11** If the corridor is not set to rebuild automatically, on the Corridor Edit Tools panel, click  Update Corridor to rebuild the corridor. Corridor surfaces and any other data that is derived from the corridor are updated using the overrides.

**To override subassembly parameters by inserting or deleting subassemblies**

**1** Select a corridor in the drawing.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor .

**3** To view detailed information about the assembly and subassemblies used by the region in the Corridor Parameters vista: On the Section Editor tab, Corridor Edit Tools panel, click  Parameter Editor.

**4** In the Section Editor tab, Station Selection panel, Select a Station list, select the station to which you want to add subassemblies.

**5** Select a subassembly from a tool palette in the Tool Palettes window.

**6** Press Enter to create a detached subassembly.

**7** To add the subassembly to the assembly: On the Corridor Edit Tools dropdown, click  Add Subassembly.

**8** In the drawing window, select the subassembly to which you want to attach the new subassembly.

**9** Optionally, if you are inserting the subassembly between two existing subassemblies, select the second assembly.

**10** To delete a subassembly: On the Corridor Edit Tools dropdown, click  Delete Subassembly.

**11** In the drawing window, click the subassembly you want to delete.

The subassembly is deleted. If the deleted subassembly connected two subassemblies, they are attached to each other at the attachment point of the deleted subassembly.

**12** To view only the stations that have assembly overrides applied to them: On the Station Selection panel, click 🖼 Show Overridden Stations. Only stations with assembly overrides are displayed in the list.

**13** To view all stations, click 🖼 Show Overridden Stations again.

**14** If the corridor is not set to rebuild automatically, on the Corridor Edit Tools panel, click 🔄 Update Corridor to rebuild the corridor. Corridor surfaces and any other data that is derived from the corridor are updated using the overrides.

---

**NOTE** You cannot undo a subassembly deletion. To rebuild the original design, insert a new instance of the subassembly or repeat the assembly selection for the region.

---

**To override subassembly parameters by editing a user-defined code**

**1** Select a corridor in the drawing.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor 🖼 .

**3** In the Section Editor tab, Station Selection panel, Select a Station list, select the station for which you want to edit feature codes.

**4** To edit a feature code: On the Corridor Edit Tools dropdown, click 🖼 Edit User Defined Codes.

**5** In the corridor section view, select a point marker, link, or shape.

**6** Do one of the following:

■ Enter a new code for the feature.

■ Enter d (delete), and then click Yes to delete the user-defined codes associated with the feature.

---

**NOTE** You can only delete user-defined point, link, and shape codes. You cannot delete codes added by the subassembly macro using the Edit User Defined Code command.

---

**7** If the corridor is not set to rebuild automatically, on the Corridor Edit Tools panel, click 🔄 Update Corridor to rebuild the corridor. Corridor surfaces and any other data that is derived from the corridor are updated using the overrides.

**To override subassembly parameters by grip editing**

**1** Select a corridor in the drawing.

**2** Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor 🖼 .

**3** In the Section Editor tab, Station Selection panel, Select a Station list, select the station at which you want to edit.

**4** Ctrl+click a subassembly in the corridor section view.
Grip behavior depends on the type of grip:

| Grip Type | Behavior |
| --- | --- |
| Square grip at end of link | Moves the end of one of more links at this point to any point in the grid. When active, the following values are labeled: offset from baseline or controlling offset, elevation, grade, and slope.To maintain the elevation value while editing, keep the point near its original vertical location. |

| Grip Type | Behavior |
|---|---|
| | To maintain the offset value, keep the point near the original horizontal location. |
| Triangular grip at end of link | Moves the endpoint of a link while preserving the grade. When active, the following values are labeled: offset and length of link, elevation difference from original location. |
| Square grip at link mid-point | Moves the link to a parallel position above or below the current position. When active, the following values are labeled: offset from baseline or controlling offset, elevation, elevation difference from original location. |

**5**  Identify the grip you want to move, and then Ctrl+click it once to make it active.

The grip changes color from blue to red, and the link property labels are displayed.

**6**  To edit a value in one of the grip labels, press Ctrl+Tab until the value you want to edit is active. Enter a new value.

**7**  Drag the grip to its new location, and click to place it there. The grip follows your mouse pointer, within the limits imposed by the grip type.

### Quick Reference

Ribbon

Corridor tab ➤ Modify panel ➤ Corridor Section Editor 

Menu

Corridors menu ➤ View/Edit Corridor Section

Command Line

ViewEditCorridorSection

Dialog Boxes

# Exporting Corridor Data

Use the corridor export utilities to export corridor data.

You can export alignments along known point codes/feature lines, feature lines for grading (using the grading feature), corridor surfaces as surface objects, and COGO points at stations on a baseline where assemblies are inserted.

# Exporting Corridor Feature Lines

Export feature lines as polylines, grading feature lines, alignments, and profiles.

**NOTE**  Exported feature lines are not connected to the corridor and are not updated if the corridor changes.

## Exporting Feature Lines as Polylines

Export polylines from corridor feature lines.

Use this option if you want to use the corridor geometry for another purpose, such as for drafting or for surface data.

**To export corridor feature lines as polylines**

1   Click Home tab ➤ Create Design panel ➤ ▼ ➤ Create Polyline From Corridor ⬚ .

2   In the drawing, click the corridor feature line. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

    The feature line is exported as a polyline. The feature line's point code is displayed at the command line.

### Quick Reference

Ribbon

  Home tab ➤ Create Design panel ➤ ▼ ➤ Create Polyline From Corridor ⬚

Command Line

  CreatePolylineFromCorrdor

Dialog Box

  Select a Feature Line (page 1838)

## Exporting Feature Lines as Grading Feature Lines

Export corridor feature lines as grading feature lines.

Use this option to create a feature line that grading commands recognize and use as a grading baseline or target.

A grading feature line represents an object in the drawing from which to grade. For more information, see Creating Feature Lines (page 743).

**To export corridor feature lines as a grading feature line**

1   Click Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line From Corridor ⬚ .

2   In the drawing, click the corridor feature line you want to export. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

3   In the Create Feature Line From Corridor dialog box:

    ■   Select the Name check box and enter a name for the feature line.

    ■   Click the Create Dynamic Link To The Corridor check box to maintain a link between the feature line and the corridor.

4   If more than one site exists in the drawing, the Pick A Site dialog box is displayed. Select a site.

5   The feature line is exported as a grading feature line. For information about grading feature lines, see Creating Feature Lines (page 743).

> **NOTE** Grading feature line objects are not named components in AutoCAD Civil. The style for the feature line is derived from the corridor object properties for the point code from which the feature line was derived. For information about changing the style for the grading feature line, see Using Grading Styles (page 737).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Feature Line drop-down ➤ Create Feature Line From Corridor

Menu

Corridors menu ➤ Utilities ➤ Create Grading Feature Line From Corridor

Command Line

FeatureLinesFromCorridor

Dialog Box

Select a Feature Line (page 1838)

# Exporting Corridor Feature Lines as Alignments

Export corridor feature lines as alignments.

Use this command if you want to apply an alignment style or label set, or to view a specific feature line in profile view.

**To export feature lines as alignments**

1  Click Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Corridor .

2  In the drawing, select a corridor feature line to export. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

3  Enter the required data in the Create Alignment - From Polyline dialog box. For more information, see Creating an Alignment from Graphic Entities (page 891).

4  Select another feature line to export as an alignment or press Enter to end the command.

5  The feature line(s) is exported as an alignment and can be viewed and edited in the Toolspace Prospector tree. For information, see Editing Alignments (page 988).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Alignment drop-down ➤ Create Alignment From Corridor

Menu

Corridors menu ➤ Utilities ➤ Create Alignment From Corridor

Command Line

CreateAlignFromCorridor

Dialog Box

Create Alignment - From Polyline (page 1760)

## Exporting Corridor Feature Lines as Profiles

Export corridor feature lines as profiles.

Profiles that are exported from corridor feature lines are listed in Prospector under the baseline alignment from which the original corridor feature line is derived.

**To export feature lines as profiles**

1 Click Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Profile From Corridor .

2 In the drawing, select the corridor feature line to export. If you make an ambiguous selection, the Select A Feature Line dialog box is displayed. Select a feature line from the list.

3 In the Create Profile - Draw New dialog box, enter the required data. For more information, see Creating Layout Profiles (page 1048).

The message "<profile name> is created" is displayed at the command line. If you have any profile views in the current drawing for the baseline alignment, the new corridor profile is added to it and displayed with the selected style.

4 Select another feature line to export as a profile or press Enter to end the command.

5 The feature line(s) is exported as a profile and can be viewed and edited in the Toolspace Prospector tree. For information, see Profile Properties (page 1024).

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Profile drop-down ➤ Create Profile From Corridor 

Menu

Corridors menu ➤ Utilities ➤ Create Profile From Corridor

Command Line

CreateProfileFromCorridor

Dialog Box

Create Profile - Draw New (page 2142)

## Exporting Corridor Points as COGO Points

Export corridor points as COGO points.

You can either export all points from a selected corridor or constrain the selection based on station ranges or point code types.

COGO points are basic building blocks in AutoCAD Civil. Use points in land development projects to identify existing ground locations and design elements.

For more information about COGO points, see Understanding Points (page 407).

**To export corridor points as COGO points**

1 Click Corridor tab ➤ Launch Pad panel ➤ ▼ ➤ Points From Corridor ⬚ .

2 In the drawing, click the corridor from which you want to export the points.

3 In the Export COGO Points (page 1825) dialog box, click For Entire Corridor Range to export all points or click For User Specified Range to select a range of points.

4 Optionally, if you selected For User Specified Range, click the Select A Baseline list to select the baseline or click ⬚ and select an alignment in the drawing.

5 If you selected For User Specified Range, enter the start and end range in the Alignment Start and Alignment End fields, or click ⬚ and click the locations in the drawing.

6 In the New Point Group Name field, enter the name for the point group that will contain all the extracted points. For information about point groups, see Point Groups (page 521).

7 Optionally, select the points you want to export, based on their point codes, by selecting or clearing the point code's Select field in the Select Point Codes To Export table. For information about point codes, see Understanding Point, Link, and Shape Codes (page 1406).

8 Click OK.

The point group is created and is displayed in the Point Groups collection in the Toolspace Prospector tree. The points are displayed in the drawing using the default Point Group display settings. They are labeled with their point code names.

## Quick Reference

Ribbon

Corridor tab ➤ Launch Pad panel ➤ ▼ ➤ Points From Corridor ⬚

Menu

Corridors menu ➤ Utilities ➤ Create COGO Points From Corridor

Command Line

CreatePointsFromCorridor

Dialog Box

Create COGO Points (page 1825)

# Exporting Corridor Surfaces

You can export corridor surfaces as detached (DTM) surface objects.

You export corridor surfaces, which are components of the corridor object, as AutoCAD Civil surface objects. For more information, see Surfaces (page 599).

The exported surface comprises breaklines based on how the corridor surface was defined (links or feature lines).

For information about corridor surfaces, see Creating and Editing Corridor Surfaces (page 1350).

**NOTE** Exported surfaces are no longer part of the corridor. They will not react to changes to the underlying corridor object.

**To export corridor surfaces as detached (DTM) surfaces**

1   Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Corridor .

2   In the drawing, click the corridor from which you want to export the surface.

    The Export Corridor Surfaces (page 1828) dialog box is displayed. View all the available corridor surfaces for the selected corridor.

3   Clear the Select field for a surface you do not want to export.

4   Optionally, to set the style for an exported surface, click the Surface Style field. Select a new style in the Pick Corridor Surface Style dialog box.

5   Optionally, to set the render material for an exported surface, click the Render Material field. Select a new render material in the Select Render Material dialog box.

6   Click OK.

    The surface(s) is created and available in the Surfaces collection in the Toolspace Prospector tree. The surfaces are displayed in accordance to the surface style set in the Export Corridor Surfaces dialog box.

## Quick Reference

Ribbon

   Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Corridor

Menu

   Corridors menu ➤ Utilities ➤ Create Detached Surfaces From Corridor

Command Line

   CorridorExtractSurfaces

Dialog Box

   Create Corridor Surfaces (page 1828)

# Rendering Corridor Models

Render corridor data using the AutoCAD Render command.

For more information about rendering AutoCAD Civil objects and data, see Rendering Objects (page 1579).

**To render corridor data**

1   Create a corridor model (page 1331) for an alignment.

2   Optionally, create feature line styles (page 1350) with different colors and apply them to the feature lines in the corridor model. For example, create a blue Edge Of Pavement style to distinguish it from a red Shoulder style.

3   Use the Feature Lines tab (page 1818) of the Corridor Properties dialog box to assign the feature line styles to the feature lines. This step helps distinguish the feature lines for easier selection when you are creating the corridor boundaries. A corridor boundary specifies an area of the corridor to render using a specified render material.

**4** Use the Surfaces tab (page 1818) of the Corridor Properties dialog box to create a corridor surface. For example, create a surface using the Links data type and the Top code. Assign a simple surface style that does not have triangles visible, such as border and contours only.

**NOTE** You can more easily create the corridor boundaries if the triangles are not visible.

**5** Select the Render Material for the corridor surface.

**6** Optionally, use the Boundaries tab (page 1820) of the Corridor Properties dialog box to create corridor boundaries that correspond to the areas on the corridor that you want to render with a specific material. You create the boundaries by selecting corridor feature lines that form the boundaries of the area. You should select the bottom end of one feature line followed by the top end of its pair.

For example, select the bottom outside edge of the lane followed by the top outside edge of the lane to create a boundary that encompasses both lanes:
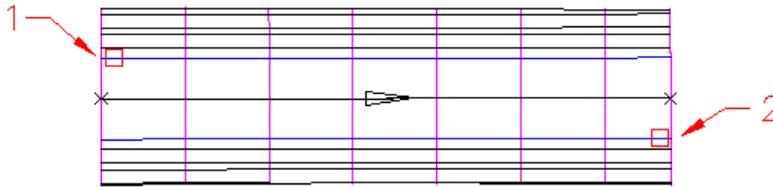


**7** Use the AutoCAD Render command to render the corridor. For more information, see Rendering Objects (page 1579).

# Corridor Command Reference

You can use these commands to quickly access corridor functionality.

The following table lists the corridor-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| CorridorExtractSurfaces | Exports a (DTM) surface object from a corridor surface (page 1373) |
| CreateAlignFromCorridor | Creates an alignment from a corridor feature line (page 1371) |
| CreateBoundaryFromCorridor | Creates a boundary at the outer edge of corridor (page 1357) |
| CreateCorridor | Creates a corridor (page 1332) |
| CreateProfileFromCorridor | Creates a profile from a corridor feature line (page 1372) |
| CreatePointsFromCorrdor | Exports corridor points as COGO points (page 1372) |
| CreatePolylineFromCorrdor | Exports a corridor feature line as a polyline (page 1370) |

| Command | Description |
| --- | --- |
| CreateSimpleCorridor | Creates a basic corridor (page 1331) |
| DefineCorridorSurfaces | Defines a surface object for the selected corridor (page 1353) |
| FeatureLineFromCorridor | Creates a grading feature line from a corridor feature line (page 1370) |
| IsolateCorridorRegion | Turns off the visibility of all corridor regions except the selected region (page 1338) |
| ShowAllCorridorRegions | Restores the visibility of all corridor region (page 1338) |
| UpdateCorridor | Rebuilds the selected corridor (page 1361) |
| ViewEditCorridorSection | Displays corridor sections (stations) (page 1362) |

# Assemblies and Subassemblies

# 31

Assembly and subassembly objects create the primary structure of an AutoCAD Civil corridor model.

## Understanding Assemblies

Assembly objects contain and manage a collection of subassemblies that are used to form the basic structure of a 3D corridor model.

Adding one or more subassembly objects, such as travel lanes, curbs, and side slopes, to an assembly baseline creates an assembly object. This forms the design for a corridor section.

The following illustration shows a simple assembly object that represents one side (lane) of a two-lane road. Subassembly objects named BasicLane and BasicCurbAndGutter have been added to a baseline alignment, forming a single travel lane with a curb and gutter.

Assembly baseline

BasicLane subassembly

BasicCurbAndGutter subassembly
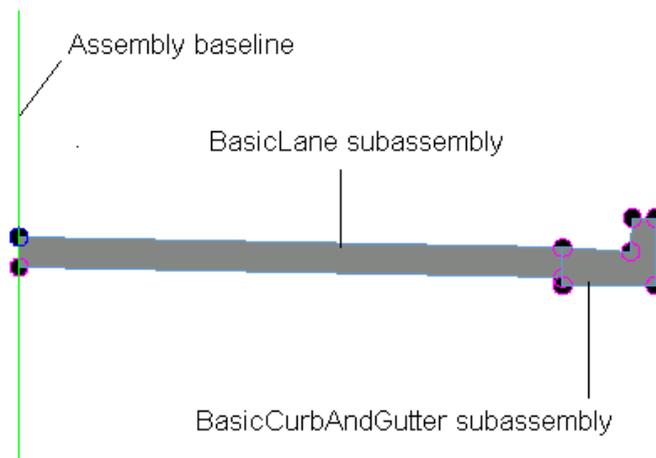
It is also possible to create more advanced assemblies referred to as conditional assemblies. A conditional assembly contains one or more conditional subassemblies, which apply subsequent subassemblies when

specified conditions at a given station are met. For more information, see Creating Assemblies with Conditional Subassemblies (page 1384).

After creating assembly objects, you can proceed with other corridor modeling tasks, such as creating corridor objects, feature lines, and section views. For more information, see Corridors (page 1327).

Review the assemblies and corridors tutorials for step-by-step instructions on how to create a variety of assemblies for corridor model.

## The Assembly Object

Use an assembly object to create the structure for a corridor section.

An assembly is a AutoCAD Civil drawing object (AECCAssembly) that manages a collection of subassembly objects. Together, assemblies and subassemblies function as the basic building blocks of a roadway or other alignment-based design. An assembly object must be applied along an alignment to form a corridor, and it can reference one or more offsets.
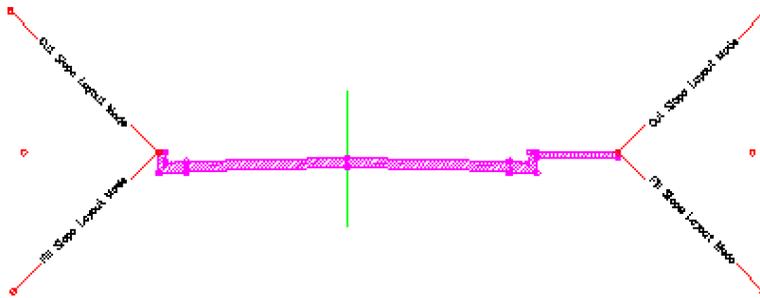


An assembly object, like its corridor and subassembly counterparts, can adapt to conditions such as superelevation and cut or fill requirements.

The following components comprise an assembly object:

■ **Insertion Point**. This is the initial point in the drawing that is selected to create the assembly object. It corresponds to the centerline of the eventual corridor object. This is also known as the ground reference point and typically follows an alignment as well as a design profile (vertical alignment).

■ **Baseline**. The baseline of an assembly typically displays as a visual aid (marker) representing a vertical axis at the assembly baseline point. If you want to attach a subassembly to the baseline point, you can do so by selecting the baseline marker. This method of attaching subassemblies to an assembly is sometimes easier than selecting the baseline point, especially when there are already one or more subassemblies attached at that point.

■ **Baseline Point**. This is a point on the assembly typically representing the start point of the first subassembly that is attached to the assembly near the controlling alignment. By default, the baseline point coincides with the insertion point and therefore follows the centerline alignment and profile. If you want to begin sectional elements oriented away (horizontally and vertically) from the centerline, do so by moving this baseline point away from the assembly insertion point.

■ **Offset Line**. The offset line is a typically vertical line visually representing a vertical axis at the offset point. If you want to attach a subassembly at an offset point, you can do so by selecting the offset line marker instead of selecting the offset point. This method of attaching subassemblies to an assembly offset point is sometimes easier than selecting the offset point, especially when there are already one or more subassemblies attached at that point.

■ **Offset Point**. This is a point on the assembly representing the ground reference point along an offset alignment for the eventual corridor object. Subassemblies attached at this point follow an offset alignment

and its designed profile. For example, in the case of a highway with service roads on one or both sides, the service road centerlines are represented by offset points. There is always just one baseline point on an assembly, and there can be zero or many offset points on an assembly. Offset points can be added to or deleted from an assembly at any time.

To complete the definition of an assembly object, you typically add multiple subassembly objects, such as lanes, curbs, or ditches, along an alignment. Each subassembly can connect to the assembly baseline point, any assembly offset point, or to another subassembly already associated with the assembly. A subassembly can also be attached to these points with a relative offset and/or elevation from the point. For more information, see Understanding Subassemblies (page 1394).

## Assembly Settings

Use assembly settings to control the behavior of assembly-related commands.

Work with assembly settings the same way you work with other object settings in AutoCAD Civil, using the Toolspace Settings tree. You can control assembly-related settings at both the object collection (feature) level and at the command level. For information about how the levels of settings work together, see Specifying Drawing Settings (page 63).

Right-click the Settings tree Assemblies collection to establish defaults for all assembly-specific settings and to override the drawing ambient settings for all assembly-related commands. Use the Commands collection in the Settings tree Assembly collection to override assembly-specific settings or drawing ambient settings for a specific command.

**NOTE** Overrides to the drawing ambient settings at the Assemblies collection level and the Assembly Commands collection level affect only the specified level. The drawing level settings are not affected. For more information, see Understanding Settings (page 61).

## Assembly Properties

Specify administrative, and codes structural information using assembly properties.

Assembly properties specify all the information associated with an assembly, including its style, subassemblies, and link, point, and shape codes.

Display the properties of an assembly:

■ Right-click an assemblies collection in the Prospector tree and click Properties.

■ Select an assembly in the drawing, right-click and click Assembly Properties.

For more information, see Assembly Properties Dialog Box (page 1779) and The Toolspace Item View (page 82).

## Assemblies Collection (Prospector Tab)

Use the Assemblies collection in the Prospector tree to access the assemblies in a drawing. As assemblies are created, they are displayed in the Assemblies collection.

Right-click the Assemblies collection to do the following:

■ Turn on or off the assembly preview (page 1391).

■ Refresh the Assemblies collection in the Prospector tree.

If an assembly has been added to the current drawing, you can expand the Assemblies collection to view the names of the assemblies in the Prospector list view. For more information, see The Toolspace Item View (page 82).

Each assembly in a site is displayed as a named object under the Assemblies node of the site. Right-click an assembly name to do the following:

- Display assembly properties (page 1379).

- Delete an assembly (page 1388) in the drawing.

- Zoom or pan to an assembly in the drawing.

- Refresh the Prospector tree.

## Assembly Collection (Settings Tab)

Use the Assembly collection in the Settings tree to manage assembly settings, styles, and command settings.

Right-click the Assembly collection to do the following:

- Edit assembly feature settings (page 1379).

- Refresh the Settings tree.

Expand the Assembly collection to display and edit the styles and command settings available for assemblies.

# Creating Assemblies

You can create an assembly from the ribbon, from a tool palette, from a catalog, or from an assembly drawing.

You can create a basic assembly, without an offset, or you can create an assembly with an offset. When you create an assembly without an offset, the subassemblies included in the assembly are controlled by a single baseline. When you add an offset to an assembly, the subassemblies included in the assembly are controlled by the baseline and by the offset alignment.

It is also possible to specify different subassemblies to apply depending on the conditions that exist at a given corridor station. This type of assembly is referred to as a conditional assembly. For more information, see Creating Assemblies with Conditional Subassemblies (page 1384).

## Creating an Assembly

Use the Create Assembly command to define the baseline of an assembly, and start adding subassemblies to the assembly.

Define the assembly basepoint (1), then add subassemblies (2,3,4) to the baseline to build the assembly (5).
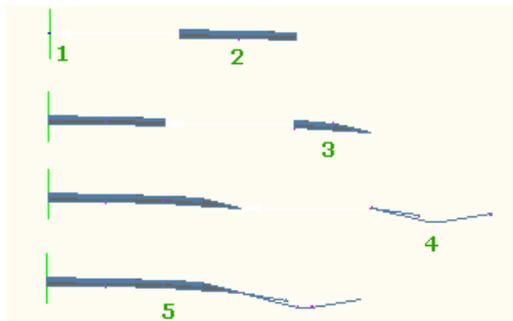
When designing an assembly that contains a left side and a right side, typically you add the subassemblies to the one side, and then add subassemblies to the other side.

The following options are available from both the Assembly and Subassembly tabs (when you click on an assembly or subassembly in the drawing):

**Copy To Assembly** lets you copy subassemblies from one assembly to another, or to another location (marker point) on the same assembly.

**Move To Assembly** lets you move subassemblies from one assembly to another, or to another location (marker point) on the same assembly.

**Mirror Subassemblies** is available on subassemblies that have a Side parameter. You can use it to quickly copy subassemblies from one side of an assembly to the other.

While creating an assembly, you can first add the right-side (right lane) subassemblies, then use Mirror Subassemblies to copy the right-side subassemblies to the left-side (left lane) of the assembly. Subassembly parameter values are automatically set to appropriate values.

Alternately, you can enter CopySubassemblyTo, MoveSubassemblyTo, or MirrorSubassembly on the Command line, to invoke the commands.

**NOTE** Corridor target settings (page 1343) of a copied, moved or mirrored subassembly are not preserved. You will have to set any targets (page 1344) of the subassembly.

**To create an assembly**

1   Click Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly  .

2   In the Create Assembly dialog box (page 1784), in the Name field, enter a name for the assembly.

   **NOTE** To name the assembly, select its default name and enter a new name, or you can use the Name Template. For more information, see Name Template Dialog Box (page 1806).

3   For Description, enter an optional description of the assembly.

4   For Assembly Style and Code Set Style, either accept the default style, select another style, or create a new style. For more information, see Assembly Styles and Display (page 1389) and Using Code Set Styles (page 1415).

5   Click  to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

   **NOTE** If you do not select a layer, the assembly is placed on the default layer.

6   Click OK.

7   To insert the assembly into the drawing, click a baseline location in the drawing.

   The assembly name is displayed under the Assemblies collection in the Prospector tree. A vertical line with a circular marker in the middle is inserted into the drawing. This is the assembly baseline location point, where you will attach one or more subassemblies.

8   Select a subassembly from a tool palette in the Tooltip Palette vista (page 1383), or from a tool catalog through the Content Browser.

   **NOTE** Not all subassemblies in AutoCAD Civil are located on a tool palette by default. All subassemblies are stored in the Content Browser, and you can add subassemblies (page 105)that you use frequently to a tool palette for easy access.

9  To attach the subassembly to the assembly baseline location, select the baseline point or the baseline marker. Typically, the baseline marker displays as a vertical line.

The subassembly name is displayed in the Subassemblies collection in the Prospector tree. A subassembly group (page 1401) is added to the assembly.

10  To add subassemblies to the subassembly just added, do any of the following:

- Select a subassembly in a tool palette, or in a tool catalog, and then select an appropriate marker point on the previously added subassembly.

- Select a subassembly that is already attached to an assembly and click ![icon] Copy To Assembly, ![icon] Move To Assembly, or ![icon] Mirror Subassemblies on the Modify Subassembly panel of the Subassembly tab. You can also select multiple subassemblies using these features.

When you add a set of subassemblies sequentially to a baseline (for example, lane, curb, sideslope, then ditch), they are all added to the same subassembly group. The next time you select the assembly baseline, a new group is created and subsequent subassemblies added belong to the new subassembly group.

For information about editing and managing assemblies, see Managing and Editing Assemblies (page 1387).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly ![icon]

Menu

Corridors menu ➤ Create Assembly

Command Line

CreateAssembly

Dialog Box

Create Assembly (page 1784)

# Creating an Assembly Offset

Specify a controlling offset within an assembly.

An assembly with an offset is useful for a corridor section that has more than one definitive path, such as a highway with a service road on one or both sides. Create the offset (3) by placing it a distance (2) from the existing assembly baseline (1).
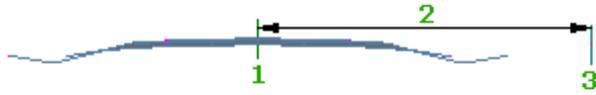
You then add subassemblies to the offset using the Subassembly Tool Palette.

The service road centerline(s) can be described by a specified offset line and offset point. The offset line is a vertical line that visually represents a secondary location that will follow an offset alignment when creating a corridor, and that subassemblies can attach to.

When assembly offsets are created, they are displayed on the Construction tab of the Assembly Properties dialog box. If desired, you can rename an offset on this tab by right-clicking the offset in the list, and selecting Rename.

You can also add or remove an offset, from the Assembly contextual menu, or the Assembly right-click menu. For more information, see Adding an Assembly Offset (page 1389) and Removing an Assembly Offset (page 1389)

**To create an assembly offset**

1  Click Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Add Assembly Offset .

2  Do one of the following:
   ■  Click an assembly in the drawing.

   ■  Press Enter to select an assembly from a list.

3  Click an offset location in the drawing.
   The offset location is displayed in the drawing.

For information, see Managing and Editing Assemblies (page 1387).

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Add Assembly Offset

Command Line

AddAssemblyOffset

# Creating an Assembly from a Tool Palette

You can create an assembly object by adding it to your drawing from a tool palette.

Once selected in the tool palette, you can add the assembly to your drawing.

**To create an assembly from a tool palette**

1   To display the Tool Palettes window:

Click Home tab ➤ Palettes panel ➤ [icon] .

2   Open the tool palette containing the desired assembly.

3   On the tool palette, click the assembly, and then click a location in the drawing. Press Enter to end the command. You can also use drag and drop.

The assembly object is added to the drawing. The assembly name is displayed under the Assemblies collection in the Prospector tree.

## Quick Reference

Ribbon

Click Home tab ➤ Palettes panel ➤ [icon] .

Command Line

ToolPalettes

# Creating Assemblies with Conditional Subassemblies

You can create an assembly that contains conditional subassemblies.

An assembly that contains one or more conditional subassemblies can be referred to as a conditional assembly. When you have a conditional assembly in your drawing, specified subassemblies are automatically applied at stations when conditions exist.

For example, a conditional corridor assembly could contain two ConditionalCutOrFill subassemblies. The first ConditionalCutOrFill subassembly can be configured to apply a ditch subassembly when a cut condition exists. The second ConditionalCutOrFill subassembly can be configured to apply a guardrail, shoulder, and daylight to surface when a fill condition exists. Using conditional subassemblies enables you to reduce the number of corridor regions and assemblies that you have to maintain.

The following illustration shows an example of an assembly that contains a ConditionalCutOrFill subassembly that is configured to apply a ditch in cut situations, and another ConditionalCutOrFill subassembly that is configured to apply a guardrail, shoulder, and to daylight to surface in fill situations.
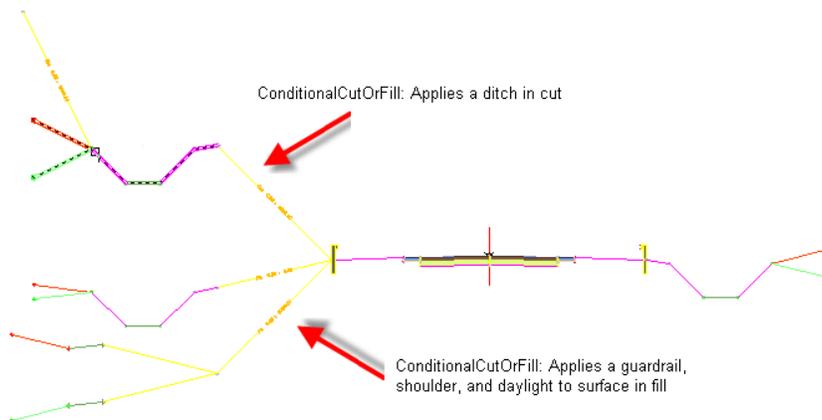
**To create a conditional assembly**

The following procedure describes how to add a conditional subassembly to an existing assembly.

1 Create an assembly containing at least one subassembly. For instructions on how to do this, refer to Creating an Assembly (page 1380).

2 In the tool palette, click the desired conditional subassembly. For example, you may want to select the ConditionalCutOrFill subassembly, which may be located on the Conditional tool palette.

3 In the Properties palette, set the subassembly's Side, Layout Width, Layout Grade, Type, and Minimum\Maximum Distance parameters to indicate the conditions under which they should be applied. For example, the table below shows how to set these parameters for the ConditionalCutOrFill subassembly in a fill situation. You may want to specify these settings differently, depending on the type of condition you want to configure.

| Parameter | Value | Description |
|---|---|---|
| Side | **Left** | Specifies which side of the assembly or corridor the subassembly is inserted towards. |
| Layout Width | **20.0000'** | Specifies the length of the line that is drawn to represent this subassembly in layout mode. This parameter, in combination with the Layout Grade parameter, allows you to position the ConditionalCutOrFill subassembly and subassemblies that are attached to it, but does not display or have any effect in the corridor model. |
| Layout Grade | **4.000:1** | Specifies the grade of the line that is drawn to represent this subassembly in layout mode. This parameter, in combination with the Layout Width parameter, allows you to position the ConditionalCutOrFill subassembly and subassemblies that are attached to it, but does not display or have any effect in the corridor model. |
| Type | **Fill** | Specifies the type of condition that when detected subsequent specified subassemblies will be applied to the assembly. |
| Minimum Distance | **0.0000'** | When the Cut or Fill (Type) depth is between these two values, the subsequent subassemblies will be applied to the assembly. |
| Maximum Distance | **5.0000'** | |

**NOTE** For the ConditionalCutOrFill subassembly, the Layout Width and Layout Grade parameters only affect the appearance of the subassembly in layout view. These parameters enable you to position the conditional subassembly and the subassemblies that are attached to it, but do not affect the corridor model.

4 If desired, you can add more conditional subassemblies to the same side of the assembly.

**5** Configure the opposite side of the assembly as desired. You can add conditional subassemblies to one or both sides.

**6** When you are finished adding subassemblies to the assembly, press Esc to exit subassembly placement mode.

The next step is to assign descriptive names to each of the subassemblies in the conditional assembly. Assigning meaningful names to the subassemblies makes it easier to identify them when you are assigning targets, and helps you to identify subassemblies in the Subassemblies collection in Prospector.

**7** For instructions on how to add meaning names to conditional subassemblies in an assembly, see Assigning Meaningful Names to Conditional Subassemblies (page 1386).

After assigning meaningful names to the conditional subassemblies, the final step is to set the corridor targets. For more information, see Setting and Editing Targets (page 1343).

**To assign meaningful names to conditional subassemblies**

**1** In the drawing, right-click the conditional assembly and select Assembly Properties.

**2** In the Assembly Properties dialog box, click the Construction tab.

Subassemblies that are added to the one side of an assembly display the default names, followed by the side to which they were added.

**3** In the Item list, select the conditional subassembly that is located on the left side of the assembly. Click it again to highlight the text. Change the name so that it indicates the type of condition it is set to detect. In the following example, the name of the first conditional subassembly has been changed to **COND Fill 0-5 for TR-L**. In this example, **COND** indicates that this is a conditional subassembly. **Fill** indicates the type of condition that activates it. **0-5** indicates the minimum and maximum distance value of the fill condition it is looking for. In this example, **TR-L** is a reference to the name of the parent assembly this particular subassembly is associated with (in this case, "Through Road" or "TR" and "Left side" or "L".)
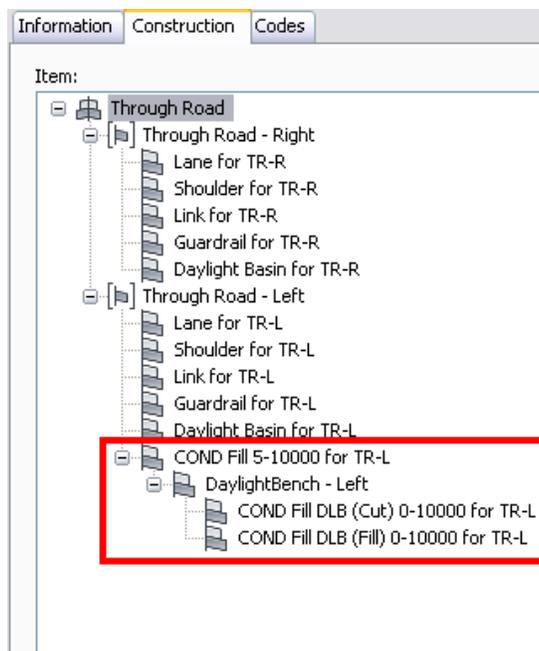


**4** Rename any other conditional subassemblies in the assembly to meaningful names.

**5** Click OK.

After assigning meaningful names to the conditional subassemblies, the final step is to set the corridor targets. For more information, see Setting and Editing Targets (page 1343).

### Quick Reference

Object Shortcut Menu

Right-click conditional assembly ➤ Properties

Dialog Box

Construction tab (Assembly Properties) (page 1779)

# Managing and Editing Assemblies

This section covers a variety of tasks you may perform while working with assemblies in AutoCAD Civil.

## Planning an Assembly

Before creating an assembly, identify the different types of subassemblies you need, make sure they are available, and decide how you want them to display.

The easiest way to access subassemblies is through a tool palette. Drag a subassembly (2) from a palette to add it to an assembly (1). You can add more subassemblies to the expanded assembly (3).
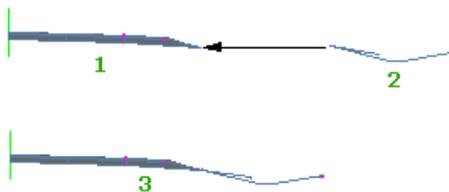


You can also add assemblies to a tool palette you create. For more information, see Understanding Subassemblies (page 1394).

You can design styles that control the appearance of the assembly components, such as the insertion point and baseline point, in the drawing. For more information, see Assembly Styles and Display (page 1389).

Use code set styles (page 1415) to control the appearance of subassembly components, such as points, links, and shapes. These components will display within your assembly object.

## Creating Assembly Sets

If you use the Create Intersection wizard to create intersections, you may want to create customized assembly set (page 2467)s for intersection creation. Assembly sets are XML files, used to organize assemblies for creating intersections. Assembly sets reference groups of existing assemblies and may reference null assemblies (page 2478). Null assemblies are used as placeholders, when complete assemblies are unavailable.

For more information on assembly sets, see Assembly Sets (page 1457).

For more information on null assemblies, see Null Assemblies (page 1460).

# Copying an Assembly

Copying an assembly creates a new assembly using the standard object naming convention.

If the assembly has subassemblies associated with it, new subassembly objects are also created. The new assembly is placed in the drawing and in the Assemblies collection in the Prospector tree. The new subassemblies are included in the assembly, and are also displayed in the Prospector tree as new subassembly objects.

When you copy an assembly, the new assembly is named using an incremental naming convention. For example if the original assembly that is copied is named "Assembly 1", the new assembly is named "Assembly 1 (1)". If assembly 1 is copied again, the new assembly is named "Assembly 1 (2)". If "Assembly 1 (1)" is copied, the new assembly is named "Assembly 1 (1) (2)".

This same naming convention also applies to any new subassemblies created when copying an assembly.

**To copy an assembly**

1 In the drawing area, right-click the assembly baseline and click Basic Modify Tools ➤ Copy.

   At the command line, you are prompted to specify a base point for the assembly.

2 Pick an insertion point in the drawing area.

   The assembly is copied in the drawing and added as a new assembly to Assemblies collection.

3 To edit the name of the assembly, right-click the assembly in the Prospector tree and click Properties.

   The Assembly Properties dialog box is displayed with the Information tab (page 1779) active. The Name field contains the name of the copied assembly, which is, by default " <assembly name> (1)."

4 Enter a new name in the Name field.

5 To edit the description of the assembly, enter a new description in the Description field.

6 Click Apply to accept the changes, or click OK to accept the changes and close the Assembly Properties dialog box.

## Quick Reference

Object Shortcut Menu

Right-click assembly baseline ➤ Basic Modify Tools ➤ Copy

Dialog Box

Information tab (Assembly Properties) (page 1779)

# Deleting an Assembly

Deleting an assembly erases it, and associated subassemblies, from the drawing and removes them from the Assemblies and Subassemblies collections in the Prospector tree.

You can delete a subassembly within an assembly, and it deletes the subassembly but not the assembly.

**To delete an assembly, do one of the following**

■ In the drawing area, right-click the assembly baseline, or select the entire assembly object, and click Basic Modify Tools ➤ Erase.

■ In the Prospector tree, right-click the assembly and click Delete.

The assembly is erased from the drawing and removed from the Assemblies collection in the Prospector tree.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Assemblies ➤ Right-click <assembly name> ➤ Delete

## Adding an Assembly Offset

You can add an offset to an assembly object.

For more information on assembly offsets, see Creating an Assembly Offset (page 1382).

**To add an assembly offset**

1 In the drawing area, right-click the assembly baseline and click Add Offset.

2 Select an offset location in the drawing.
   The offset location is displayed in the drawing.

### Quick Reference

Object Shortcut Menu

Right-click assembly baseline ➤ Add Offset

## Removing an Assembly Offset

You can remove an offset from an assembly which erases the offset from the drawing.

For more information on assembly offsets, see Creating an Assembly Offset (page 1382).

**To remove an assembly offset**

1 In the drawing area, select the assembly baseline from which you want to remove the offset.

2 Right-click and click Remove Offset.

If there are multiple offsets associated with the assembly, you are prompted to select an offset. If there is only one offset associated with the assembly, it is removed.

### Quick Reference

Object Shortcut Menu

Right-click assembly baseline ➤ Remove Offset

## Assembly Styles and Display

Use assembly styles and code set styles to control the visual display of each assembly component.

# Using Assembly Styles

Use styles to control the visual appearance of the assembly object components.

You can create specific styles to use for the different phases of a project. For example, you can create one style to use in the design layout phase and another style to use for plotting.

Use assembly styles to control the appearance of assembly baseline location, profile grade reference point, controlling offset baseline location and profile point markers.

You can access assembly styles using the methods in the following table:

| To access assembly styles... | Then... |
| --- | --- |
| In the drawing | Right-click an assembly and then click Edit Assembly Style. |
| On the Settings tab in Toolspace | Right-click a style in the Assembly Styles folder and then click Edit. |

**NOTE** When you click New on the shortcut menu of a collection folder, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a new style from an existing style, right-click the style, click copy, and then save the style with a new name.

Use the various tabs in the Assembly Style dialog box to set the following style information:

■ **Information**. Specifies name, description, and creation information.

■ **Marker**. Specifies the display of marker symbols (such as for the baseline point) within an assembly.

■ **Display**. Lists the assembly components and display options as well as the choice of 2D or 3D visibility. You can display components in a 2D plan view or a 3D view using other tools, such as Orbit or Vpoint.

■ **Summary**. Lists all the style properties.

# Copying or Editing Assembly Styles

Use the Settings tree to copy or edit an assembly style by clicking an existing style, making modifications and then saving it with a new name.

**To copy or edit an assembly style**

1 In Toolspace, on the Settings tab, expand the Assembly Styles collection and right-click an existing style.

2 Click Copy or Edit.

3 In the Assembly Style dialog box, click the Information (page 1781) tab and enter a new name and description for the assembly style.

4 To specify the display of marker symbols (such as for the baseline, baseline point, and insertion point), click the Marker (page 1781) tab and specify the marker properties for the assembly.

5 To define the display properties for the assembly style, click the Display (page 1782) tab and specify the display properties for the assembly.

6 To view a summary of information about the style, click the Summary (page 1783) tab.

7 Click Apply.

### Quick Reference

Toolspace Shortcut Menu

  Settings tab: Right-click Assembly Style item

Object Shortcut Menu

  Right-click Assembly object ➤ Copy or Edit Assembly Style

Dialog Box

  Assembly Style (page 1781)

## Previewing an Assembly

You can preview an assembly in the Prospector Item List View.

**To preview an assembly**

1   In Toolspace, on the Prospector tab, right-click the Assemblies collection and click Show Preview.

> **NOTE** A check mark is displayed next to the Show Preview menu item when the preview is enabled.

2   In the Prospector tree, click the assembly that you want to preview.
    The assembly is displayed in the Prospector item view.

> **NOTE** The preview of the assembly uses the same display styles as the assembly in the drawing area.

### Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Right-click Assemblies ➤ Show Preview

## Sharing Assemblies

You can access and share assemblies using tool palettes, catalogs, or assembly drawings.

For example, you can create an assembly in your drawing, and then drag and drop it to a tool palette. Once it is in a tool palette, you can copy an assembly from a tool palette into a catalog. You can create a tool palette or a catalog that contains multiple assemblies, and then share that tool palette or catalog with others.

You can also save an assembly to a drawing file (.dwg), and then drag and drop that drawing file into your current drawing.

The following list summarizes ways you can share assemblies:

■   copy an assembly from the drawing area to a tool palette

■   copy an assembly from a tool palette to the drawing area

■   copy an assembly from a tool palette to a catalog

■   copy an assembly from a catalog to the drawing area

■   copy an assembly from a catalog to a tool palette

■   drag (copy) an assembly drawing (.dwg) into another drawing

**NOTE** You cannot copy an assembly directly from a drawing into a catalog. You must first copy the assembly to a tool palette. From a tool palette, you can copy an assembly to a catalog.

**To copy an assembly from the drawing area to a tool palette**

1 Display the Tool Palettes window.

 Click Home tab ➤ Palettes panel ➤ ![icon] .

2 Open the desired tool palette or create a new tool palette.

3 In the drawing area, zoom to the desired assembly.

4 Select the desired assembly, and click and hold down the mouse button to drag it onto the tool palette.

 **NOTE** To select an assembly, make sure that you click only on the assembly baseline (vertical line). Do not click on any grips on the assembly baseline.

The assembly is copied to the tool palette.

**To copy an assembly from a tool palette to the drawing area**

1 Display the Tool Palettes window.

 Click Home tab ➤ Palettes panel ➤ ![icon] .

2 Open the tool palette that contains the desired assembly.

3 In the tool palette, click the desired assembly, and then click a location in the drawing area.

The assembly is copied to the drawing area.

**To copy assemblies from a tool palette to a catalog**

1 To display catalogs:

 Click View tab ➤ Palettes panel ➤ ![icon] .

 **TIP** When working with both the AutoCAD Civil workspace and the Content Browser, you may find it useful to have the Content Browser window displayed at all times. To do this, right-click in the Content Browser title bar and select Always On Top.

2 To display tool palettes: Click Palettes panel ➤ ![icon] .

3 Open the tool palette that contains the desired assembly or assemblies.

4 In the tool palette, click on the desired assembly or assemblies, and hold down the mouse button to drag and drop them into the desired catalog.

The assemblies are copied to the catalog.

**To copy an assembly from a catalog to the drawing area**

You can copy an assembly from a catalog directly into the drawing area.

1 To display catalogs in the Content Browser: Click View tab ➤ Palettes panel ➤ ![icon] .

**2** Navigate to the catalog containing the desired assembly.

**3** Once you have located the desired assembly, click on the i-drop icon ⬤ that is located on the assembly image. When you do this the cursor image changes to an eye-dropper ⬖ .

**4** Drag the eyedropper icon into the drawing area.

The assembly is copied into the drawing area.

### To copy an assembly from a catalog to a tool palette

You can copy an assembly from a catalog directly onto a tool palette.

**1** To display catalogs in the Content Browser: Click View tab ➤ Palettes panel ➤ 🔲 .

**2** Navigate to the catalog containing the desired assembly.

**3** Once you have located the desired assembly, click on the i-drop icon ⬤ that is located on the assembly image. When you do this the cursor image changes to an eye-dropper ⬖ .

**4** Drag the eyedropper icon into the desired tool palette.

The assembly is copied into the tool palette.

### To drag (copy) an assembly from a drawing (.dwg) to another drawing

You can drag and drop an assembly drawing into another drawing. In this scenario, the assembly drawing that you want to copy (drag) must contain just the assembly that you want to copy, and no other data.

**1** Using Windows Explorer, navigate to the folder containing the assembly drawing file.

**2** Right-click on the assembly drawing file and drag and drop it into the AutoCAD Civil drawing area.

The assembly contained in the assembly drawing file is copied into the currently open AutoCAD Civil drawing.

The assembly is inserted as a block. Explode it to create the assembly object. When the assembly object is created, it appears in the Assemblies node in Prospector.

## Quick Reference

Ribbon

Click Home tab ➤ Palettes panel ➤ 📱

Click View tab ➤ Palettes panel ➤ 📱

Click View tab ➤ Palettes panel ➤ 🔳

Command Line

ContentBrowser
ToolPalettes

# Understanding Subassemblies

Subassemblies are the basic building blocks of a corridor design.

A subassembly is an AutoCAD drawing object (AECCSubassembly) that defines the geometry of a component used in a corridor section. Through the tool palette and tool catalogs, AutoCAD Civil provides preconfigured subassemblies for components such as travel lanes, curbs, side slopes and ditches. These subassemblies are defined by a set of points, links, and optionally closed areas referred to as shapes.

The subassemblies provided with AutoCAD Civil have built-in intelligent behavior. They can automatically adapt to conditions such as superelevation and cut or fill requirements. For example, a side slope subassembly has variable slopes that change automatically depending on the depth of cut along the corridor. In fill conditions exceeding a given depth, the shoulder automatically widens to include a guardrail or barrier.

After you determine the kinds of subassemblies to use, make sure they are available through an AutoCAD Civil tool palette or tool catalogs available through the Content Browser. For more information, see the Content Browser Help.

In addition to ready-to-use subassemblies, AutoCAD Civil also lets you create your own custom subassembly objects from polylines, or design more sophisticated ones using a .NET language. (See *AutoCAD Civil Developer's Guide*.)

### Layout and Modeling Modes

A subassembly that has not yet been applied to a corridor is considered to be in layout mode. After you create a corridor, all subassemblies included in the corridor model are considered to be in modeling mode.

### Conditional Subassemblies

You can use a special type of subassembly called a conditional subassembly that automatically inserts other specified subassemblies when certain conditions are met.

For example, a conditional corridor assembly could contain two ConditionalCutOrFill subassemblies. The first ConditionalCutOrFill subassembly can be configured to apply a ditch subassembly when a cut condition exists. The second ConditionalCutOrFill subassembly can be configured to apply a guardrail, shoulder, and daylight to surface when a fill condition exists. Using conditional subassemblies enables you to reduce the number of corridor regions and assemblies that you have to maintain.

An assembly that contains one or more conditional subassemblies can be referred to as a conditional assembly. For more information, see Creating Assemblies with Conditional Subassemblies (page 1384), or the tutorial exercise Creating an Assembly with Conditions.

# The Subassembly Object

A subassembly is an AutoCAD Civil drawing object (AECCSubassembly) that defines the geometry of a component used in a corridor section.

Add subassembly objects to an assembly object to create a road cross section. Through tool palette and tool catalogs, AutoCAD Civil provides preconfigured subassemblies for components such as travel lanes, curbs, side slopes and ditches.

For information on stock subassemblies, see Subassemblies at a Glance.

# Subassembly Styles

Use code set styles to control the appearance of subassemblies.

The subassembly object itself does not have its own style. However, since you will typically be working with sets of subassemblies, you can assign styles to subassembly components (point, links, and shapes) through code set styles. For more information, see Using Codes and Code Set Styles (page 1405).

# Subassembly Settings

Use subassembly settings to control the behavior of subassembly-related commands.

Work with subassembly settings the same way you work with other object settings in AutoCAD Civil, using the Toolspace Settings tree. You can control subassembly-related settings at both the object collection (feature) level and at the command level. For information about how the levels of settings work together, see Specifying Drawing Settings (page 63).

Right-click the Settings tree Subassembly collection to establish defaults for all subassembly-specific settings and to override the drawing ambient settings for all subassembly-related commands. Use the Commands collection in the Settings tree Subassembly collection to override subassembly-specific settings or drawing ambient settings for a specific command.

**NOTE** Overrides to the drawing ambient settings at the Subassembly collection level and the Subassembly Commands collection level affect only the specified level. The drawing level settings are not affected. For more information, see Understanding Settings (page 61).

# Subassembly Properties

Specify administrative, parameter and codes information using subassembly properties.

Subassembly properties specify all the information associated with a subassembly, including its code set style, input and output parameters, the name of the macro file it is derived from, and a list of the link, point, and shape codes associated with it.

Display the properties of a subassembly:

■ Right-click a subassemblies collection in the Prospector tree and click Properties.

■ Select a subassembly in the drawing, right-click and click Subassembly Properties.

For more information, see Subassembly Properties Dialog Box (page 1790) and The Toolspace Item View (page 82).

# Subassemblies Collection (Prospector Tab)

Use the Subassemblies collections in the Prospector tree to access the subassemblies in a drawing.

Right-click the Subassemblies collection to do the following:

■ Turn on or off the subassembly preview (page 1402).

■ Refresh the Subassemblies collection in the Prospector tree.

If a subassembly has been added to the current drawing, you can expand the Subassemblies collection to view the names of the subassemblies in the Prospector list view. For more information, see The Toolspace Item View (page 82).

Each subassembly in a site is displayed as a named object under the Subassemblies node of the site. Right-click a subassembly name to do the following:

■ Display subassembly properties (page 1395).

■ Delete the subassembly (page 1401) in the drawing.

■ Zoom or pan to the subassembly in the drawing.

■ Refresh the Prospector tree.

## Subassembly Collection (Settings Tab)

Use the Subassembly collection in the Settings tree to manage subassembly settings and commands.

Right-click the Subassembly collection to do the following:

■ Edit subassembly feature settings (page 1395).

■ Refresh the Settings tree.

Expand the Subassembly collection to display and edit the subassembly commands.

# Creating Subassemblies

Add subassemblies to a drawing using the subassemblies provided through the AutoCAD Civil tool palette or tool catalog, or create your own custom subassemblies.

When using the preconfigured AutoCAD Civil subassemblies, adding the subassembly to the drawing creates the subassembly object.

If you want to create a custom shape and use it as a subassembly, use the Create Subassembly from Polyline feature. For more information, see Creating Subassemblies From Polylines (page 1397).

Another approach to creating customized subassemblies is using scripts in a .NET language. This method provides you with the ability to create custom subassemblies with simple or complex conditional behavior built in. For more information, see Creating Custom Subassemblies Using .NET in the *AutoCAD Civil Developer's Guide*.

# Creating a Subassembly from a Tool Palette

You can create a subassembly object by adding it to your drawing from a tool palette.

Once selected in the tool palette, you can add the subassembly to an assembly baseline, or to a subassembly attached to an assembly. You can also just add the subassembly to your drawing and not associate it with an assembly. This is referred to as a detached subassembly.

**To create a subassembly from a tool palette**

1  Display the Tool Palettes window.

   Click Home tab ➤ Palettes panel ➤ [  ] .

2  On the Tool Palettes window, click the desired subassembly.

3  Do one of the following:
   ■ To add the subassembly to an assembly, select a marker point on an assembly in the drawing.

■ To add the subassembly to the drawing, and not associate it with an assembly, press Enter then click a location in the drawing.

The subassembly name is displayed under the Subassemblies collection in the Prospector tree.

If you subsequently move the subassembly from its original insertion point in an assembly, you can return it to that point by clicking Clear Offset in Assembly on the Modify Subassembly panel, or by entering SubassemblyClearOffset at the Command line.

The Clear Offset in Assembly button is only enabled if a selected subassembly is part of an assembly, and was moved away from its original insertion point.

### Quick Reference

Ribbon

Click Home tab ➤ Palettes panel ➤ 

Command Line

ToolPalettes
CreateSubAssemblyTool
SubassemblyClearOffset

## Creating Subassemblies From Polylines

You can create customized subassemblies from AutoCAD polyline objects in a drawing.

To create a subassembly from a polyline object, first create the desired polyline object in the drawing. Add the new subassembly to an assembly by right-clicking it and clicking Add To Assembly.

Make sure the polyline object is free of drawing errors.

After you have created the subassembly from a polyline, add the desired point, link, and shape codes to the subassembly. If you do not add any codes to the subassembly, the subassembly does not display when you create the corridor. For more information, see Understanding Point, Link, and Shape Codes (page 1406) and Adding Codes to a Subassembly (page 1414).

Use code set styles (page 1405) to control the visual appearance of subassembly components. You can use existing code set styles or create your own.

**To create a subassembly from a polyline**

1 Create the desired polyline shape in the drawing.

2 Click Home tab ➤ Create Design panel ➤ ▼ ➤ Create Subassembly From Polyline .

3 Click the polyline object in the drawing and press Enter.

4 In the Create Assembly (page 1784) dialog box, for Name, enter a name for the subassembly.

   **NOTE** To name the subassembly, you can select its default name and enter a new name, or you can use the Name Template. For more information, see Name Template Dialog Box (page 1806).

5 For Description, enter an optional description for the subassembly.

6 In the Code Set Style list, select a Code Set Style or accept the default. For more information, see Using Codes and Code Set Styles (page 1405).

**7** Click ![layer icon] to select a layer. For more information about layers, see Object Layer Dialog Box (page 1985).

> **NOTE** If you do not select a layer, the subassembly is placed on the default layer.

**8** Optionally, select a mid-ordinate distance.

If this subassembly contains curves, they will be tessellated (converted into a finite number of segments). This value determines the length of the tessellated segments.

**9** In the Link Creation list, choose the behavior for link creation by selecting Multiple, Single, or None. Multiple is the default.

Multiple: Creates a new link for each segment created from the object selected. This is useful for assigning link codes to different components of the subassembly for creating surfaces, or other display features.

Single: Creates one link from all segments created from the selected object. This is provided to make creating shapes simpler. Using the Add Shape right-click command, you only need to perform one action.

None: Produces no links; only points will be created at the vertices.

**10** Optionally, you may clear Erase Existing Entities if you want to save the original polyline object selected to create the subassembly. By default, this option is checked and the polyline selected is automatically deleted.

**11** Click OK. The new subassembly is created and is displayed in the Subassemblies collection in the Prospector tree.

To add codes to the subassembly, see Adding Codes to a Subassembly (page 1414).

To add the subassembly to an assembly, see Creating Assemblies (page 1380).

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ ▼ ➤ Create Subassembly From Polyline ![icon]

Menu

Corridors menu ➤ Create Subassembly From Polyline

Command Line

CreateSubFromPline

Dialog Box

Create Subassembly From Polyline (page 1784)

## Inserting Subassemblies on Right and Left Sides of a Corridor

Some subassemblies use a Side input parameter to specify inserting the subassembly on the right or left side of a corridor; other subassemblies allow you to do this by specifying a positive or negative value for the insertion point input parameter.

Subassemblies that contain a Side input parameter let you specify which side the subassembly is inserted by setting this parameter to Right or Left. By default, this parameter is typically set to Right. Change it to Left to insert the subassembly onto the left side.

Other subassemblies, such as LinkOffsetAndElevation, do not contain a Side input parameter. However, you can insert the subassembly onto the right or left side of a controlling baseline by entering a positive or a negative value for the insertion point input parameters. A positive value for this parameter inserts the subassembly onto the right side of a corridor, or of a controlling baseline. A negative value inserts the subassembly onto the left side. The insertion point input parameter could be named any of the following, depending on the function of the subassembly: Insert Point Offset, Sample Point Offset, Daylight Offset from Baseline.

The following are examples of subassemblies that do not contain an input parameter named Side. For these types of subassemblies, you specify the side by using positive and negative values for the insertion point input parameter.

■ DaylightToOffset

■ DaylightToRow

■ LinkOffsetAndElevation

■ LinkOffsetAndSlope

■ LinkOffsetOnSurface

■ OverlayMillAndLevel1

■ OverlayWidenMatchSlope1

■ OverlayWidenWithSuper

You can also use the Mirror option to automatically copy a subassembly, or multiple subassemblies, from one side of an assembly to the other. For more information, see Creating an Assembly (page 1380)

**To insert a subassembly on the left side of a corridor using the Side parameter**

1  Click Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly 🏗 .

2  In the Create Assembly (page 1784) dialog box, accept the default settings. Click OK.

3  When you are prompted to specify the assembly baseline location, click a point in the drawing to insert the assembly.

4  In the Tool Palettes window, open the Imperial - Basic tool palette by clicking the Imperial - Basic tab. (To display the Tool Palettes window, click Home tab ➤ Palettes panel ➤ 🗔 .)

5  Click the BasicLane subassembly. The Properties dialog box is displayed for the BasicLane subassembly. Note that the Side parameter is set to Right by default.

6  Click on the assembly. The BasicLane subassembly is inserted onto the right side of the assembly.

7  On the tool palette, click the BasicLane subassembly again. Again the Properties dialog box is displayed.

8  In the Properties dialog box, in the Advanced section, click the value for the Side parameter and change it from Right to Left.

9  Click on the assembly. The BasicLane subassembly is inserted on the left side of the assembly.

**To insert a subassembly on the left side of a corridor using (negative) insertion point values**

1  Click Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly 🏗 .

2  In the Create Assembly dialog box, accept the default settings. Click OK.

**3** When you are prompted to specify the assembly baseline location, click a point in the drawing to insert the assembly.

**4** In the Tool Palettes window, open the Imperial - Daylight tool palette by clicking the Imperial - Daylight tab. (To display the Tool Palette window, click Home tab ➤ Palettes panel ➤ ⬚ .)

**5** Click the DaylightToOffset subassembly. The Properties dialog box is displayed for the DaylightToOffset subassembly. Note that the Daylight Offset from Baseline parameter is set to 10 by default.

**6** Click on the assembly. The DaylightToOffset subassembly is inserted onto the right side of the assembly.

**7** On the tool palette, click the DaylightToOffset subassembly again. Again the Properties dialog box is displayed.

**8** In the Properties dialog box, in the Advanced section, click the value for the Daylight Offset from Baseline parameter and change it from positive 10 to negative 10.

**9** Click on the assembly. The DaylightToOffset subassembly is inserted on the left side of the assembly.

**To insert a subassembly on the left side of a corridor using Mirror**

**1** Click Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly 🏭 .

**2** In the Create Assembly (page 1784) dialog box, accept the default settings. Click OK.

**3** When you are prompted to specify the assembly baseline location, click a point in the drawing to insert the assembly.

**4** In the Tool Palettes window, open the Imperial - Basic tool palette by clicking the Imperial - Basic tab. (To display the Tool Palette window, click Home tab ➤ Palettes panel ➤ ⬚ .)

**5** Click the BasicLane subassembly. The Properties dialog box is displayed for the BasicLane subassembly. Note that the Side parameter is set to Right by default.

**6** Click on the assembly. The BasicLane subassembly is inserted onto the right side of the assembly.

**7** On the assembly, right-click on the BasicLane subassembly and click Mirror. The BasicLane subassembly is inserted on the left side of the assembly. Note that the Mirror option is only available for subassemblies that contain a Side parameter.

## Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Assembly drop-down ➤ Create Assembly 🏭

Home tab ➤ Palettes panel ➤ ⬚

Menu

Corridors menu ➤ Create Assembly

Command Line

CreateAssembly
ToolPalettes

Dialog Box

Create Assembly (page 1784)

# Managing and Editing Subassemblies

There are a variety of ways you can work with subassemblies in AutoCAD Civil.

## Deleting a Subassembly

Deleting a subassembly erases it from the drawing and removes it from the Subassemblies collection in the Prospector tree.

When you delete an assembly, and subassemblies associated with that assembly are also deleted.

**To delete a subassembly, do one of the following**

- In the drawing area, right-click the subassembly and click Basic Modify Tools ➤ Erase.

- In the Prospector tree, right-click the subassembly and click Delete.

The subassembly is erased from the drawing and removed from the Subassemblies collection in the Prospector tree. If the subassembly was attached to an assembly, it is removed from the assembly too.

### Quick Reference

Object Shortcut Menu

   Right-click subassembly ➤ Basic Modify Tools ➤ Erase

Toolspace Shortcut Menu

   Prospector tab: Subassemblies ➤ Right-click <subassembly name> ➤ Delete

## Managing Subassembly Groups

Each subassembly added to an assembly is automatically associated with a named group within the assembly.

Subassembly groups manage the order in which subassemblies are processed during corridor modeling. The first time you add a subassembly to an assembly, the subassembly is added to the first group. When you add a second subassembly by attaching it to the first subassembly, the second subassembly also gets added to the first subassembly group. The next time you select an assembly baseline, a new subassembly group is automatically created and subsequent subassemblies added to the assembly are added that group.

The Construction tab (page 1779) of the Assembly Properties dialog box displays the subassembly groups for a given assembly. The location of the groups on this tab reflects the order in which the subassemblies are processed. The group of subassemblies located at the very top of the list is processed first, in top-to-bottom order. The group of subassemblies located directly beneath the top group is processed second, and so on.

**NOTE** Subassembly groups are not processed according to the numeric order associated with their subassembly group name. For example, a subassembly group named Group (1) located at the bottom of the Item list is processed last, even though the group is named Group (1).

You can change the order of subassembly groups by moving them up or down. However, you cannot reorder subassemblies within a group. If you delete subassemblies or subassembly groups using this tab, the subassemblies are deleted from the drawing.

Subassemblies that are not associated with assemblies (detached subassemblies and subassemblies created from polylines) are not associated with groups.

**To move a subassembly group**

**1** In Toolspace, on the Prospector tab, expand the Assemblies collection and right-click an existing assembly.

**2** Click Properties.

**3** Click the Construction tab.

**4** In the Item list, right-click the name of a subassembly group and select Move up or Move down.

The group of subassemblies is moved up or down in the list.

## Previewing a Subassembly

You can preview a subassembly in the Prospector Item List View.

**To preview a subassembly**

**1** In Toolspace, on the Prospector tab, right-click the Subassemblies collection and click Show Preview.

**NOTE** A check mark is displayed next to the Show Preview menu item when the preview is enabled.

**2** In the Prospector tree, click the subassembly you want to preview.

The subassembly is displayed in the Prospector item view.

**NOTE** The preview of the subassembly uses the same display styles as the subassembly in the drawing area.

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Right-click Subassemblies ➤ Show Preview

## Converting VBA Subassemblies to .NET

You can convert AutoCAD Civil stock subassemblies that are VBA-based to .NET.

In Civil 3D 2007, and in earlier versions of Civil 3D, the subassemblies that were included in the product were created using VBA.

In Civil 3D 2008, all of the subassemblies included in the product were created using .NET. This is because .NET versions of subassemblies provide faster performance than VBA-based subassemblies when generating corridor models. Civil 3D 2008, and later versions, still support VBA-based subassemblies. For example, in drawings, catalogs, and tool palettes, you can have a mix of VBA-based and .NET-based subassemblies. However, it is recommended that you convert VBA-based subassemblies to .NET using the following procedure to achieve optimal performance when generating corridor models.

**NOTE** This command only converts the VBA-based AutoCAD Civil stock subassemblies that were included with the product. It does not convert any VBA-based subassemblies that were created from user-defined (custom) VBA code.

To determine if a subassembly is VBA-based, .NET-based, or created from a polyline, use the Subassembly Properties dialog box or the Properties palette to display the subassembly definition.

For information on creating custom subassemblies, see the AutoCAD Civil Developer's Guide.

**To display the subassembly definition in the Subassembly Properties dialog box**

1 Right-click a subassembly in the drawing and click Subassembly Properties.

2 In the Subassembly Properties dialog box, click the Parameters tab.

The Defined From field displays .NET if the subassembly was created using .NET, VBA if it was created using VBA, and Entities if the subassembly was created from a polyline.

**To display the subassembly definition in the Properties palette**

1 Right-click a subassembly in the drawing and click Properties.

2 On the Properties palette, click the Design tab.

The Geometry Generate Mode property displays .NET if the subassembly was created using .NET, VBA if it was created using VBA, and Entities if the subassembly was created from a polyline.

**To convert subassemblies from VBA to .NET**

1 Click Modify tab ➤ Design panel ➤ ▼ ➤ Convert VBA Subassemblies To .NET ⬛ .

2 Do one of the following:

■ To convert all subassemblies in the drawing, at the command line, enter **A**, and then press Enter. All VBA subassemblies in the drawing are converted to VB.NET.

■ To convert all subassemblies in a selected assembly, select an assembly, and then press Enter. All VBA subassemblies in the selected assembly are converted to VB.NET.

## Quick Reference

Ribbon

Modify tab ➤ Design panel ➤ ▼ ➤ Convert VBA Subassemblies To .NET ⬛

Menu

Corridors menu ➤ Utilities ➤ Convert VBA Subassemblies To .NET

Object Shortcut Menu

Right-click subassembly ➤ Subassembly Properties
Right-click subassembly ➤ Properties

Toolspace Shortcut Menu

Prospector tab: Subassemblies ➤ Right-click <subassembly name> ➤ Properties

Command Line

ConvertVBASAsToDotNET

Dialog Box

Parameters tab (Subassembly Properties) (page 1790)

# Sharing Subassemblies

You can access and share subassemblies using tool palettes, the AutoCAD Civil Corridor Modeling catalogs, or package files.

For example, you can create custom subassemblies in your drawing, and then drag and drop them onto a tool palette or catalog. You can create a tool palette or catalog that contains multiple custom subassemblies and then share that tool palette or catalog with others.

Custom subassemblies that have been created using .NET or VBA can be exported and imported using a package file. Subassemblies created from polylines cannot be included in a package file.

A package file contains all the files necessary for the custom subassemblies to work. Once the package file is created, users can import the package file contents into AutoCAD Civil using the ImportSubassemblies command, and copy the custom subassemblies directly into a tool palette or catalog. You can also share a package file with others. For example, you could attach a package file containing one or more subassemblies to an e-mail, or use other file transfer methods.

For more information on creating package files, see Exporting Custom Subassemblies Using a Package File in the *AutoCAD Civil Developer's Guide*.

For information on adding subassemblies to tool palettes, see Tool Palettes (page 105).

**To copy (import) subassemblies from a package file to a tool palette**

1   To display the Tool Palettes window:

    Click Home tab ➤ Palettes panel ➤ 　 .

2   Open the tool palette that you want to copy the subassemblies into. You may open an existing tool palette or create a new one.

3   Click Insert tab ➤ Import panel ➤ ▼ ➤ Import Subassemblies 　 .

4   On the Import Subassemblies (page 1787) dialog box, click the folder icon to navigate to and select the package file.

5   In the Import To section, click Tool Palette, and then select the desired tool palette in the drop-down list.

    **NOTE** You can import the subassemblies to a tool palette and to a catalog at the same time. When you import subassemblies to a catalog, they automatically get imported to the My Imported Tools catalog. If the My Imported Tools catalog does not exist, it is automatically created.

6   Click OK.

The subassemblies contained in the package file are copied to the tool palette. If you chose to copy the subassemblies to a catalog at the same time, the subassemblies are also copied to the My Imported Tools catalog.

**NOTE** Subassemblies created from polylines cannot be included in a package file. Package files are intended for sharing custom subassemblies that have been created using .NET or VBA.

**To copy (import) subassemblies from a package file to a catalog**

1   To display the corridor modeling catalogs:

    Click View tab ➤ Palettes panel ➤ 　 .

    **TIP** When working with both the AutoCAD Civil workspace and the Content Browser, you may find it useful to have the Content Browser window displayed at all times. To do this, right-click in the Content Browser title bar and select Always On Top.

2   Click Insert tab ➤ Import panel ➤ ▼ ➤ Import Subassemblies 　 .

**3** On the Import Subassemblies (page 1787) dialog box, click the folder icon to navigate to and select the package file.

**4** In the Import To section, click Catalog Library/My Imported Tools.

> **NOTE** You can import the subassemblies to a tool palette and to a catalog at the same time. When you import subassemblies to a catalog, they automatically get imported to the My Imported Tools catalog. If the My Imported Tools catalog does not exist, it is automatically created.

**5** Click OK.

The subassemblies contained in the package file are copied to the My Imported Tools catalog.

> **NOTE** When you import subassemblies to a catalog using this method (package files), you can only copy the subassemblies to the My Imported Tools catalog. If the My Imported Tools catalog does not exist, one is automatically created. Once the subassemblies are in the My Imported Tools catalog, you can then copy them to other catalogs.

If you chose to copy the subassemblies to a tool palette at the same time, the subassemblies are also copied to the selected tool palette.

> **NOTE** Subassemblies created from polylines cannot be included in a package file. Package files are intended for sharing custom subassemblies that have been created using .NET or VBA.

### Quick Reference

Ribbon

Insert tab ➤ Import panel ➤ ▼ ➤ Import Subassemblies 🖼️

Menu

Corridors menu ➤ Utilities ➤ Import Subassemblies

Tool Palette Shortcut Menu

In Tool Palette: Right-click Import Subassemblies

Command Line

ImportTools

Dialog Box

Import Subassemblies (page 1787)

# Using Codes and Code Set Styles

Codes can be associated with the point, link, and shape components of a subassembly.

Codes allow you to apply simple or complex rules governing the offset and elevation behavior of subassemblies. You can choose to add codes to certain subassembly components to implement presentation or annotation features to a corridor design.

A typical subassembly is comprised of the following components, which can have codes associated with them:

- **Point.** Define the basic subassembly structure.

- **Link.** Connect points to form planar structural surfaces of a corridor.

- **Shape**. Defined from links and define a closed region that represents materials used in a corridor model.

For a set of point, link or shape components, you can define a set of display characteristics and assign them to a code set style. You can then assign a code set style to a subassembly, assembly, corridor, or section view.

You can assign appropriate label styles to various code sets in the code set styles.

## Pavement Structure of Paved Sections

The AutoCAD Civil Corridor Modeling subassemblies that model paved components of a roadway, such as travel lanes and shoulders, allow multiple material layers to be defined.

These material layer sections are defined by sets of links. The typical structure is shown in the following diagram.

**Pavement Structure on Paved Sections**
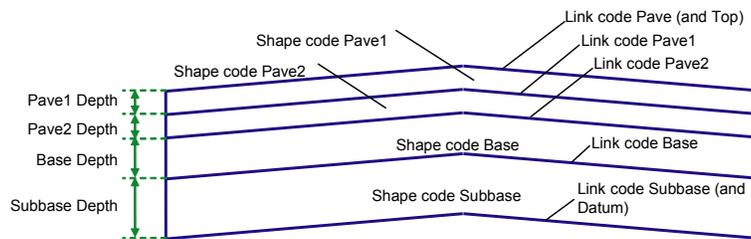


The standard link codes from top to bottom are Pave, Pave1, Pave2, Base, and Subbase. The code Top is also assigned to all finish grade links, thus coinciding with the Pave links on paved sections. The code Datum is also assigned to the bottom of subbase. The shape codes for the material areas from top to bottom are Pave1, Pave2, Base, and Subbase.

The subassemblies that create paved sections have the depth or thickness of each layer as user-definable input parameters. Each subassembly that creates paved sections closes the layer areas to form a closed shape, even if those layers actually continue into the adjacent component. Layers can be omitted by setting the corresponding depth to zero. This collapses that layer so that the shape also has a zero area.

## Understanding Point, Link, and Shape Codes

The optional codes associated with a subassembly point, link, or shape components determine some of the behavior you will see in corridor modeling.

When designing custom subassemblies, you must consider not only the geometric shape of the corridor model that will ultimately be created by the subassemblies, but also how the model is used for post-modeling design operations. These operations include:

- Design surface modeling

- Combining the corridor model with grading models

- Earthwork and material volume analysis

- Design visualization and rendering

- Right-of-way acquisition

- Slopestake reporting

- Export to survey data recorders for construction stakeout

- Plans production

- Staged construction modeling

Nearly all of these operations depend on the codes assigned to points, links, and shapes in the corridor model. To create models that integrate into all of the operations listed above, the coding scheme must be carefully designed before creating the first subassembly. The best practice is to diagram all of the different roadway types and situations that can be anticipated for the users you are supporting. Next, devise and annotate codes for all critical points, links, and shapes. Use this diagram when documenting the codes for each subassembly. Note that it is difficult to design a single set of codes that meet needs of every operation. For this reason, AutoCAD Civil lets you assign any number of codes to each point, link, and shape. How these codes are utilized is described in the following sections.

### Creating Point, Link, and Shape Codes

You can create new point, link, and shape codes using a .NET language.

For more information about creating point, link, and shape codes using .NET languages, see *AutoCAD Civil Developer's Guide.*

For more information about assigning code set styles to point, link, and shape codes, see Using Codes and Code Set Styles (page 1405).

### Point Codes

Point codes are the feature codes assigned to the endpoints of the links that make up the subassembly component

| Operation | Description |
|---|---|
| Corridor Modeling | The corridor model is represented graphically by roadway cross sections at predetermined stations, with longitudinal strings connecting points between adjacent stations. Points with the same code and in the same region of the section are automatically connected with longitudinal lines. |
| Plans Production | Many organizations require that critical points be annotated on cross section drawings. For example, a given state's Department of Transportation may require the following: 1.) Elevations at ditch bottoms, 2.) Slopes (x:1) of ditch sides, clear zones, and cut/fill daylight links, 3.) % Slopes of lanes and shoulders, 4.) Offsets of the cut/fill daylight points. To create custom cross-section sheet utilities that meet these requirements, these points must be identifiable within the model. Another plans production operation is depicting the limits-of-construction in plan view, with different symbology and line types used for cut lines and fill lines. |
| Design Surface Modeling and Staged Design | Point codes make it possible to extract finish grade profiles at critical points on the corridor model surfaces. For example, a profile connecting the left edge-of-pavement points of a roadway can be used in drainage analysis and design, tying links from an adjacent road's model, and to combine corridor models with grading models. |

| Operation | Description |
| --- | --- |
| Right-of-Way Analysis | The limits-of-construction of the corridor model can be determined by extracting alignments or figures that connect the cut/fill daylight points on each side of the model. These are used to determine what property has to be acquired. |
| Slopestake reporting and construction stakeout | Identifying critical points on the different layers of the corridor model is essential to creating construction staking reports, or when exporting data to survey data recorders for construction stakeout. Since grading is nearly always done to the subbase line, not to the finish grade, it may be necessary to define distinct codes on both the finish grade and subbase surface links. For example, the edge-of-traveled-way may use code ETW on finish grade, and ETW_Sub on the subbase layer. |

The cut/fill daylight points are a good example of where two codes may be needed for the same point. If the daylight points for both cut and fill links have the code "Daylight", then it is easy to depict and extract a single figure defining the limit of construction on each side. If the same points have the code "Daylight_Cut" or "Daylight_Fill," depending on circumstances, it is also easy to annotate cut and fill daylight lines differently.

### Link Codes

Link codes are the feature codes assigned to each of the links that make up the roadway component. A link is defined as a single straight-line segment between endpoints.

| Operation | Description |
| --- | --- |
| Design surface modeling | Design surfaces can be built by extracting all links with a particular code. Link codes should be designed so that it is easy to extract the types of surfaces needed for surface modeling. Typically these are used for earthwork calculations, additional grading, visualization, and drainage analysis. Depending on the application, you may need to extract paved surfaces, all finish grade surfaces, or subbase surfaces. |
| Earthwork and material volume analysis | Different layers of the corridor model can be separated and used for volume calculations using tables that define materials in terms of which link codes are above, below, left, and right of the material layers. Simple cut and fill quantities are usually determined by comparing the existing ground to a combination of unpaved finish grade links and subbase links. Pavement layers, such as wearing course, asphalt base, and granular subbase, are determined based on the surface links defining those as closed areas. |
| Visualization and rendering | For realistic renderings of design surfaces, it is useful to distinguish between different surface materials such as pavement, gravel, grass, and concrete. |

To meet the requirements of these operations, it is usually necessary to assign multiple codes to links on certain portions of the roadway. A typical scheme:

| Link Code | Description |
| --- | --- |
| Top | Assigned to all links on the finished grade, whether paved or unpaved |
| Pave | A second code assigned to all paved links on the finished grade |
| Datum | Assigned to all unpaved finished grade links, and all links on the bottoms and sides of the subbase layers |
| Subbase | Assigned to all links on the bottoms and sides of the subbase layers |

Thus, the Top links can be used to build a surface model of the entire finished grade surface, which is used for visualization and drainage analysis. The Pave links can be used to render the paved portions of the finished grade surface with a particular color or texture. The Datum links can be used to model the surface that is graded with earth moving equipment, and is also used for simple cut and fill volume analysis. The Subbase links are used to define the bottom of the granular base material areas for material volume analysis.

### Shape Codes

Shapes are closed cross-sectional areas created by a single subassembly. The primary use for shape codes; defining hatch patterns for different materials, and to extract areas for material volume tabulation. The codes should reflect how you want the materials identified in the earthwork reports.

# Standard Point Codes

A set of standard codes define the display and behavior of the point components used in the AutoCAD Civil Corridor Modeling subassemblies.

The following illustration shows the location of the standard point codes on subassemblies defining a divided road with a depressed median and truncated shoulders.
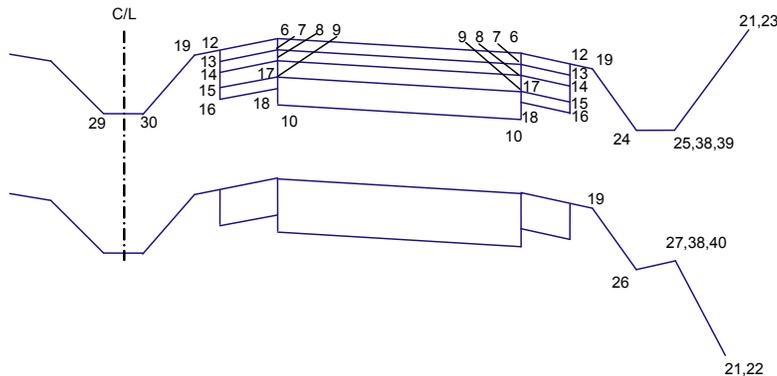


**Divided Road with Depressed Median; Truncated Shoulders**

The table following the illustrations provides descriptions of the point codes used in the illustrated assemblies.

The following illustration shows the location of the standard point codes on subassemblies defining a divided road with a depressed median and extended shoulders:
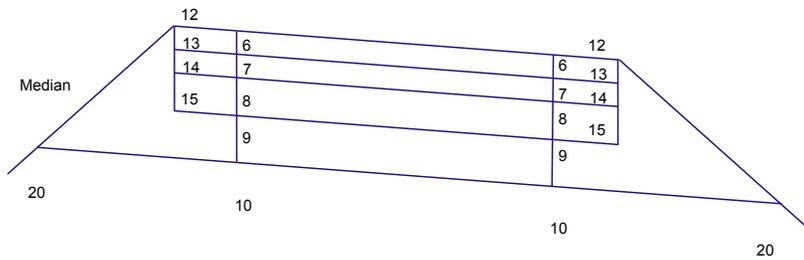
**Divided Road with Depressed Median; Extended Shoulders**

The following illustration shows the location of the standard point codes on subassemblies defining an undivided road with a curb-and-gutter. Not all layers are shown.



**Undivided Road with Curb-and-Gutter; Not All Layers Shown**

The table below names and describes the standard point codes used in the example assembly illustrations.

**Point Codes**

| No. | Code | Description |
| --- | --- | --- |
| 1 | Crown | Crown point between travel lanes on finish grade. |
| 4 | Crown_Base | Crown point between travel lanes on the base layer. |
| 5 | Crown_Sub | Crown point between travel lanes on the subbase layer. |
| 6 | ETW | Edge of traveled way; inside or outside edges of travel lanes on finish grade. |
| 7 | ETW_Pave1 | Edge of traveled way on the Pave1 layer. |
| 8 | ETW_Pave2 | Edge of traveled way on the Pave2 layer. |
| 9 | ETW_Base | Edge of traveled way on the base layer. |
| 10 | ETW_Sub | Edge of traveled way on the subbase layer. |
| 12 | Lane_Pave1 | Lane break point on Pave1. |
| 13 | Lane_Pave2 | Lane break point on Pave2. |
| 14 | Lane_Base | Lane break point on Base. |
| 15 | Lane_Sub | Lane break point on Subbase. |

| No. | Code | Description |
| --- | --- | --- |
| 16 | EPS | Edge of paved shoulder; outer edge of paved portions of shoulder on finish grade. |
| 17 | EPS_Pave1 | Edge of paved shoulder on the Pave1 layer. |
| 18 | EPS_Pave2 | Edge of paved shoulder on the Pave2 layer. |
| 19 | EPS_Base | Edge of paved shoulder on the Base layer. |
| 20 | EPS_Sub | Edge of paved shoulder; outer edge of paved portions of shoulder on the Subbase layer. |
| 21 | EPS_Base_In | Inside edge of paved shoulder on the base layer. |
| 22 | EPS_Sub_In | Inside edge of paved shoulder on the subbase layer. |
| 23 | ES_Unpaved | Edge of gravel shoulder; outer edge of unpaved portions of shoulder on finish grade. |
| 24 | Daylight_Sub | Subgrade intercept point; point where the subbase surface extends and intersects the finish grade. |
| 25 | Daylight | Daylight point for a cut or fill slope. |
| 26 | Daylight_Fill | Daylight point for a fill slope. |
| 27 | Daylight_Cut | Daylight point for a cut slope. |
| 29 | Ditch_Out | Outside edge of ditch. |
| 30 | Bench_In | Inside edge of bench. |
| 31 | Bench_Out | Outside edge of bench. |
| 32 | Flowline_Ditch | Flowline of a V-shaped ditch. |
| 33 | LMedDitch | Left edge of median ditch. |
| 38 | Bottom_Curb | Bottom of curb for a curb without gutter. |
| 39 | Back_Curb | Back of curb. |
| 40 | Sidewalk_In | Inside edge of sidewalk. |

To view all actual code numbers, the code names and brief code descriptions: In Notepad, open ..\Documents and Settings\All Users\Application Data\Autodesk\C3D2010\enu\C3DStockSubassemblyScripts.codes.

## Standard Link Codes

A set of standard codes define the display and behavior of the link components used in the AutoCAD Civil Corridor Modeling subassemblies.

The following illustration shows the location of the standard link codes on subassemblies defining a divided road with a depressed median and truncated shoulders.
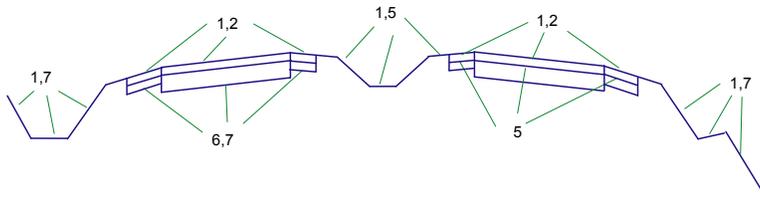
**Divided Road with Depressed Median; Truncated Shoulders**

The table following the illustrations provides descriptions of the link codes used in the illustrated assemblies.

The following illustration shows the location of the standard link codes on subassemblies defining a divided road with a depressed median and extended shoulders.
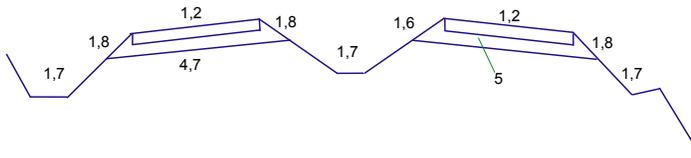


**Divided Road with Depressed Median; Extended Shoulders**

The following illustration shows the location of the standard link codes on subassemblies defining an undivided road with a curb-and-gutter. Not all layers are shown.



**Undivided Road with Curb-and-Gutter; Not All Layers Shown**

The table below names and describes the standard link codes used in the example assembly illustrations.

**Link Codes**

| No. | Code | Description |
|---|---|---|
| 1 | Top | Traces all links on finished grade, including: <br> Unpaved finish grade links (clear zone, cut/fill slopes) <br> Top paved surface on paved sections. <br> Gutter, face of curb, and top of curb. <br> Top of sidewalk. <br> For most roadways, the Top links trace the finished grade surface from the left catch point to the right catch point. These links would build the finish grade surface. |
| 2 | Pave | Any finished grade surface on a paved section. |
| 5 | Base | All links on the base surface of paved sections. |

| No. | Code | Description |
|-----|------|-------------|
| 6 | SubBase | All links on the subbase surface of paved sections. |
| 7 | Datum | All finish grade links on unpaved sections, and all subbase links on paved sections. For most roadways, the Datum links trace the unpaved finished grade and subbase from the left catch point to the right catch point. These links would build the surface used to calculate cut and fill volumes. |
| 8 | Gravel | Finish grade surfaces above gravel shoulders with subbase. |
| 12 | Curb | All links comprising a curb or curb and gutter. |
| 13 | Sidewalk | All links comprising a sidewalk. |

To view all actual code numbers, the code names and brief code descriptions: In Notepad, open ..\Documents and Settings\All Users\Application Data\Autodesk\C3D2010\enu\C3DStockSubassemblyScripts.codes.

## Standard Shape Codes

A set of standard codes define the display and behavior of the shape components used in the AutoCAD Civil Corridor Modeling subassemblies.

The following illustration shows the location of some standard shape codes on subassemblies defining an undivided road with a curb-and-gutter:
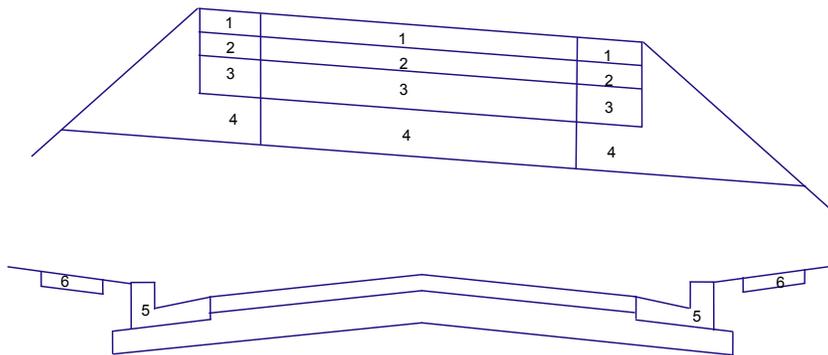


**Undivided Road with Curb-and-Gutter**

The table below names and describes the standard shape codes used in the example assembly illustration.

**Shape Codes**

| No. | Code | Description |
|-----|------|-------------|
| 1 | Pave1 | Closed shape between the finish grade and first pavement surface on paved sections. |
| 2 | Pave2 | Closed shape between the first and second pavement surface on paved sections. |
| 3 | Base | Closed shape between the second pavement surface and the base surface on paved sections. |

| No. | Code | Description |
|---|---|---|
| 4 | SubBase | Closed shape between the base and subbase surfaces on paved sections. |
| 5 | Curb | Closed area within a concrete curb or curb and gutter. |
| 6 | Sidewalk | Closed area within a concrete sidewalk. |

To view all actual code numbers, the code names and brief code descriptions: In Notepad, open ..\Documents and Settings\All Users\Application Data\Autodesk\C3D2010\enu\C3DStockSubassemblyScripts.codes.

## Adding Codes to a Subassembly

Add codes to a subassembly created from a polyline.

After you create a subassembly from a polyline, the next step is to add the necessary codes to the subassembly. For more information, see Creating Subassemblies From Polylines (page 1397).

**To add a code to a subassembly created from a polyline**

1 In the drawing, select the subassembly, then right-click it and click Add Code.

2 Enter the code and press Enter.

> **NOTE** The code you enter must be an existing code, or, if it is a new code, you must make sure that it gets added to the codes file (*C3DStockSubassemblyScripts.codes*). For more information, see Adding a New Code to the AutoCAD Civil Stock Subassemblies (page 1419).

3 Select a point, link, or shape on the subassembly.

4 To see the code, in Toolspace, on the Prospector tab, right-click the subassembly and select Properties. Click the Codes tab. The code added is displayed in the Name column under the point, link, or shape collections.

**To remove codes from a subassembly created from a polyline**

1 In the drawing, select the subassembly from which you want to remove the code.

2 Click Subassembly tab ➤ Modify Subassembly panel ➤ Remove Code .

3 Click the point, link, or shape on the subassembly.
 The code is removed from the subassembly.

### Quick Reference

Ribbon

Subassembly tab ➤ Modify Subassembly panel ➤ Remove Code

Object Shortcut Menu

Right-click subassembly object ➤ Add Code

Command Line

AddSubassemblyCode
RemoveSubassemblyCodes

# Using Code Set Styles

Use code set styles to control the visual appearance of a code set (link, point, and shape codes). You can create specific styles to use for the different aspects of an assembly or corridor.

Since subassemblies can be displayed in a number of objects, code set styles are applicable to those objects as well. Below are the objects code set styles can be associated with:

■ Subassemblies

■ Assemblies

■ Corridors

■ Section views

An individual subassembly typically contains just a few points, links and shapes. Therefore, most subassemblies typically contain a relatively simple code list. Since an assembly manages a group of subassemblies, the code list for an assembly is comparatively larger. Furthermore, since a corridor can have multiple assemblies applied within it, the code list becomes larger and complex. The same holds true for the section views.

## Adding or Importing Codes to a Code Set Style

Add or import codes to a code set style. Both methods use the Codes tab of the Code Set Style dialog box. Click the Add button to add a new or existing code, or use the Import Codes button to import codes from a subassembly, assembly, or corridor object.

**To add a code to a code set style**

1   In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Code Set Styles collection, and right-click an existing style.

2   Click either Copy or Edit.

3   In the Code Set Style dialog box, click the Codes (page 1783) tab.

4   Right-click a code collection (one of Link, Point, or Shape) in the Name column. Click Add.

5   In the Pick Style dialog box, select a style for the code.

6   In the Name column, click NEW CODE, enter the name of the code you want to add, and press Enter.

7   Click Apply.

**To import codes to a code set style**

1   In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Code Set Styles collection, and right-click an existing style.

2   Click Copy or Edit.

3   In the Code Set Style dialog box, click the Codes (page 1783) tab.

4   Click Import Codes.

5   In the drawing, click on a subassembly, assembly, or corridor object, and press Enter. The codes are automatically displayed in the Codes tab list.

6   Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: General collection ➤ Multipurpose Styles collection ➤ Code Set Styles collection ➤ Right-click <style name>

Dialog Box

Codes tab (Code Set Style) (page 1783)

## Copying or Editing Code Set Styles

Use the Settings tree to copy or edit a code set style by clicking an existing style, making modifications and then saving it with a new name.

**To copy or edit a code set style**

1   In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Code Set Styles collection, and right-click an existing style.

2   Click Copy or Edit.

3   In the Code Set Style dialog box, click the Information (page 1783) tab and enter a new name and description for the code set style.

4   Click the Codes (page 1783) tab to define styles for each link, point (marker), or shape code associated with this code set.

5   Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <code set style item>

Dialog Box

Code Set Style (page 1783)

## Copying or Editing Link Styles

Use the Settings tree to copy or edit a link style by clicking an existing style, making modifications and then saving it with a new name.

**To copy or edit a link style for subassembly links**

1   In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Link Styles collection, and right-click an existing style.

2   Click Copy or Edit.

3   In the Link Style dialog box, click the Information (page 1788) tab and enter a new name and description for the link style.

4   To define the display properties for the link style, click the Display (page 1788) tab and specify the display properties for the link.

5   To view a summary of information about the style, click the Summary (page 1788) tab.

**6**  Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <link style item>.

Dialog Box

[Link Style](#) (page 1787)

## Copying or Editing Marker Styles

Use the Settings tree to copy or edit a marker style for subassembly points by clicking an existing style, making modifications and then saving it with a new name.

**To copy or edit a marker style for subassembly points**

**1**  In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Marker Styles collection, and right-click an existing style.

**2**  Click Copy or Edit.

**3**  In the Marker Style dialog box, click the [Information Tab (Marker Style Dialog Box)](#) (page 1788)tab and enter a new name and description for the marker style.

**4**  To define the symbol used to display markers, click the [Marker](#) (page 1788) tab and specify the type of symbol, as well as the size, scaling and rotation, for the marker symbol.

**5**  To define the display properties for the marker style, click the [Display](#) (page 1789) tab and specify the display properties for the marker.

**6**  To view a summary of information about the style, click the [Summary](#) (page 1789) tab.

**7**  Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <marker style item>

Dialog Box

[Marker Style](#) (page 1788)

## Copying or Editing Shape Styles

Use the Settings tree to copy or edit a shape style by clicking an existing style, making modifications and then saving it with a new name.

**To copy or edit a shape style for subassembly shapes**

**1**  In Toolspace, on the Settings tab, expand the General collection, the Multipurpose Styles collection, then the Shape Styles collection, and right-click an existing style.

**2**  Click Copy or Edit.

**3** In the Shape Style dialog box, click the Information (page 1790) tab and enter a new name and description for the shape style.

**4** To define the display properties for the shape style, click the Display (page 1790) tab and specify the display properties for the shape.

**5** To view a summary of information about the style, click the Summary (page 1790) tab.

**6** Click Apply.


## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <shape style item>

Dialog Box

Shape Style (page 1790)


# Localizing or Modifying Subassembly Codes

For localization purposes, the AutoCAD Civil stock subassemblies use an external codes file to define the point, link, and shape codes referenced in the subassemblies.

The external codes file is named C3DStockSubassemblyScripts.codes and is located in the same folder as the C3DStockSubassemblyScripts.dvb file and/or C3DStockSubassemblies.dll file. By default, on English-language based systems, this location is typically C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu. For non-English-language-based systems, the "enu" location corresponds to the appropriate language or region.

The codes file is associated with each subassembly with the CodesSpecific module in the .dvb file. The following sections explain the definition of the code files and how to edit it or add point, link, and shape codes to it.

The CodesSpecific module is in the CodesSpecific.vb file of the C3DStockSubassemblies VB project in the case of .NET subassemblies, and in the .dvb file in the case of VBA subassemblies.


### Understanding the C3DStockSubassemblyScripts.codes File

You can open the C3DStockSubassemblyScripts.Codes file with any text editor. Add comments in this file by adding // in front of a sentence. Do not leave any blank lines in the file. Each code line is separated with a return character. Each code definition is comprised of the following components, separated by commas:

■ Index Number: An integer value used (for a point, link, or shape as required) to assign a label to the subassembly module.

■ Localized Code: The value that will be displayed AutoCAD Civil in your Subassembly, Assembly, and or Corridor Properties Codes tab, and is used to set the style for the link.

■ Original Code: The code variable used by AutoCAD Civil. No matter what localized version of the subassembly is used, it is interpreted the same way to make drawings portable across different language versions.

■ Code Description: A localized description to further explain the functionality associated with this code. This text is displayed in the Description column of the Codes tab on the subassembly, assembly, and or corridor properties dialog box. This text is optional.

**NOTE** Codes files must be saved in Unicode format. For example, if you are using Notepad to update the codes file, when you save the file, the Save As dialog box lets you choose an encoding method. Make sure you choose Unicode.

The following is an example of a code definition line in the C3DStockSubassemblyScripts.codes file:

```
38,BottomCurb,BottomCurb,Bottom of curb for a curb without gutter.
```

The Index number is: `38`

The Localized Code is: `BottomCurb`

The Original Code is: `BottomCurb`

The Code Description is: `Bottom of curb for a curb without gutter.`

**Modifying Default Codes for AutoCAD Civil Stock Subassemblies**

To change or localize code display names, open the C3DStockSubassemblyScripts.codes file in a text editor and change the Localized Code and or corresponding code description.

For example, the display name of the Curb Bottom can be changed by editing the following bolded components in the C3DStockSubassemblyScripts.codes file as follows:

38,BottomKerb,BottomCurb,Bottom of kerb for a kerb without gutter

**Adding a New Code to the AutoCAD Civil Stock Subassemblies**

If you want to modify existing subassemblies to new codes, or add a new subassembly module, and you would like to use the same structure as default content, for .NET subassemblies you can do the following:

1 Add `<New Code>` at the end of the C3DStockSubassemblyScripts.Codes file, with a new index, localized name, original code, and optional description.

2 In the CodesSpecific.vb of the C3DStockSubassemblies VB project, add the following at the end of the AllCodes structure: `Public <New Code> as CodeType`

3 In the CodesSpecific.vb of the `With/End With` block of the `FillCodeStructure` routine, add the following at the end: `GetFromCollection colCodesAndDescriptionHashtable, n, .<NewCode>`

4 Declare the default value, in case the codes file is not found, for the `CodesDefault(<New Code Index>)` in InitializeDefaults() function of the `CodesSpecific` module.

5 Change the size of array `CodesDefault` in the `CodesSpecific` module in the `Dim` statement.

A new code added using these procedures can be used for any existing or new .NET-coded subassemblies in AutoCAD Civil Stock Subassemblies.

For VBA subassemblies, you can do the following:

1 Add `<New Code>` at the end of the C3DStockSubassemblyScripts.Codes file, with a new index, localized name, and optional description.

2 In the CodesSpecific module of C3DStockSubassemblyScripts.dvb file, add the following at the end of the AllCodes structure: `g_s<New Code> as CodeType`

3 In the CodesSpecific file, in the `With/End With` block of the `FillCodeStructure` routine, add the following at the end: `GetFromCollection colCodesAndDescriptionHashtable, n, .g_s<NewCode>`

4 Declare the default value, in case the codes file is not found, for the `g_arrDefaults(<New Code Index>)` in InitializeDefaults() function of the `CodesSpecific` module.

5 Change the size of array `g_arrDefault` in the `CodesSpecific` module.

A new code added using these procedures can be used for any existing or new VBA-coded subassemblies in AutoCAD Civil Stock Subassemblies.

**NOTE** The process of associating point codes is primarily done for easy localization of default content. If you are writing custom subassemblies, you do not have to follow this procedure. You can create a new module that contains a customized list of codes your subassemblies can reference. You can also bypass this process by hard-coding strings locally within each module.

**Localizing Subassembly Catalogs and Tool Palettes**

The default subassemblies are generally defined for a wide range of clients and geographical regions. You may want to change some of the display names of catalogs, palettes, tools, or tool variables as applicable to various situations and languages. These catalogs and tool palettes are composed in XML format, in *.atc* files, and can be edited.

**NOTE** By default, on English-language-based systems, the AutoCAD Civil subassembly tool catalog *.atc* files are installed to C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Tool Catalogs\Road Catalog.

The following list identifies the data element values you can change in the subassembly tool catalog *.atc* file for localization purposes:

- The `<ItemName>` value is the descriptive name of the catalog, category, palette, or subassembly.

- The `<Description>` value describes the catalog, category, palette, or subassembly.

- The `<Keywords>` value contains a help string for subassemblies.

- The following attributes of the `<Params>` data element:
    - DisplayName: Name displayed in the AutoCAD Properties window for a selected subassembly.

    - Description: Descriptive text that is displayed at the lower left corner of the AutoCAD Properties window for a selected subassembly.

- The following data element values can be changed if needed:
    - The parameter values (such as Width in BasicLane subassembly) for any sub-element in `<Params>`.

    - The `<Units>` value which supports feet and meters in AutoCAD Civil.

    - The `<DotNetClass Assembly = ...>` (for .NET subassemblies), which can be modified to point to a different .NET class.

    - The `<Macro Module = ...>` (for VBA subassemblies), which can be modified to point to a different macro.

**NOTE** Only the values of a data element should be changed, not the data element name itself (such as Width in the BasicLane subassembly). These data elements are global variable names that are used in subassembly scripts.

**Installing Country Kits, Customized Codes Files, or Locale-specific Codes Files**

If you plan to install a country kit that contains a region-specific codes file, or, if you plan to install (use) a codes file that contains region-specific content, you must do the following before installing the country kit, or before installing the customized codes file.

Make a backup copy of the region-specific or customized codes file, rename it to the default name of your installed codes file (for example, for English version products, the default codes file is named C3DStockSubassemblyScripts_US.codes) and copy it to the following location: C:\Documents and Settings\All

Users\Application Data\Autodesk\Civil2010\enu. For non-English-language based systems, the "enu" location should correspond to the appropriate locale.

Note that when AutoCAD Civil is first installed on a new machine, the following two versions of the codes file are installed:

■ C3DStockSubassemblyScripts.codes

■ C3DStockSubassemblyScripts_US.codes

The C3DStockSubassemblyScripts_<locale>.codes file (in this example, C3DStockSubassemblyScripts_US.codes) is installed to provide a backup mechanism for securing codes file content.

When a country kit that contains a region-specific codes file is installed, the localized codes file is installed to C3DStockSubassemblyScripts.codes overwriting the existing file, and a copy is made at C3DStockSubassemblyScripts_<locale>.codes. If you have modified your C3DStockSubassemblyScripts.codes file, this is why it is important to make a backup of it before installing the country kit, or locale-specific codes file.

### Troubleshooting Codes File Issues

If you are experiencing unexpected results when using customized subassemblies, or when performing certain AutoCAD Civil corridor modeling tutorials, this may be due to the drawing attempting to reference codes file information that it cannot find. In some cases, these issues can be resolved by updating your existing codes file so that it contains the necessary information.

# Assembly and Subassembly Command Reference

You can use these commands to quickly access assembly and subassembly functionality.

The following table lists the assembly- and subassembly-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AddAssemblyOffset | Adds an offset assembly to an existing assembly (page 1383) |
| AddSubassemblyCode | Adds a code to a subassembly created from a polyline (page 1414) |
| ContentBrowser | Displays the Content Browser (page 106) |
| ConvertVBASAsToDotNET | Converts VBA subassemblies to .NET (page 1260) |
| CopySubssemblyTo | Copies a subassembly to or within an assembly (page 1380) |
| CreateAssembly | Creates an assembly (page 1381) |
| CreateSubAssemblyTool | Creates a subassembly from a tool palette (page 1396) |
| CreateSubFromPline | Creates a subassembly from a polyline (page 1397) |
| ImportTools | Imports subassemblies to a palette or catalog (page 1403) |
| MirrorSubssembly | Mirrors the selected subassembly or subassemblies within an assembly (page 1380) |

| Command | Description |
| --- | --- |
| MoveSubssemblyTo | Moves a subassembly to or within an assembly (page 1380) |
| RemoveSubassemblyCodes | Removes a code from a subassembly created from a polyline (page 1414) |
| SubassemblyClearOffset | Returns a moved subassembly to its insertion point in an assembly (page 1396) |
| ToolPalettes | Displays the Tool Palettes window (page 104) |

# Intersections

# 32

You can use AutoCAD Civil intersection design features to automatically create flexible 3D models of corridor intersections.

⟳ Intersection Design Workflow

## Understanding Intersections

An intersection object builds on and manages various object data, including alignments, profiles, corridors, assemblies, subassemblies, and surfaces.

In its simplest form, an intersection object can be created by selecting the location where two alignments intersect. This automatically creates an intersection object in the drawing, and this intersection object is also displayed in the Prospector tree.
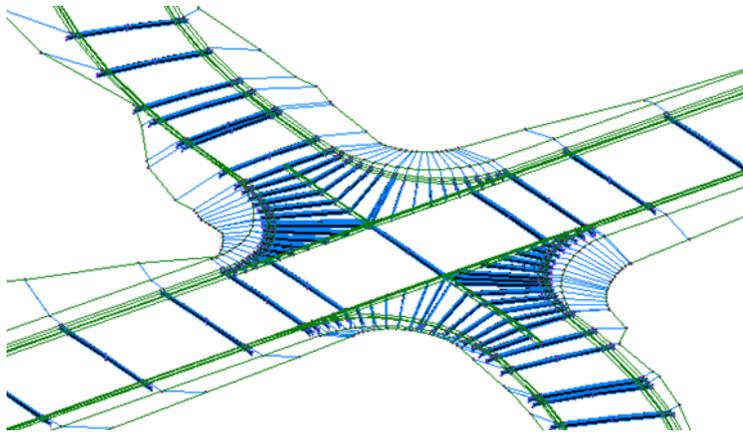


Another approach to creating intersections is to first create roadway alignments, offset alignments, and profiles. Once these preliminary design items are in place, you can create an intersection object that utilizes the 3D aspects of these road design geometry components.

Intersections are created from and based on existing AutoCAD Civil objects, which include:

■ **Alignments** (horizontal). Used by an intersection to define the centerlines of the two roads that intersect. Three types of alignments are used in intersections: centerline alignments, offset alignments, and curb return alignments. The centerline and offset alignments are used to define the horizontal geometry aspects of an intersection object.

- **Profiles** (vertical alignments). Used to define surface and design elevations along a horizontal alignment.

- **Surfaces**. Used to build surface profiles, and for corridor grading in an intersection.

- **Corridors**. A corridor can be built automatically to represent the 3D aspects of the intersection. You can also add to this new corridor information to an existing corridor on the intersecting alignment(s).

- **Assemblies**. Roadway assemblies are required to build a corridor representing the intersection. To build an intersection corridor, you must be able to access appropriate assemblies to build the intersection model. AutoCAD Civil comes with a set of default intersection assemblies that you can use, or you can create your own custom assemblies that have been designed specifically to suit the needs of intersections you create.

- **Assembly set**. An intersection corridor requires an *assembly set*, which is a set of assemblies that are designed specifically for a specific type of intersection design. The assemblies included in an assembly set are defined in an .xml file that is referred to as the assembly set file. A default assembly set and assembly set file is included with AutoCAD Civil. You can create an intersection using this default assembly set, or you can create your own custom assembly sets.

- **Subassemblies**. Subassemblies define the geometry of a corridor section (through an assembly). For example, a typical roadway may be composed of paved lanes on either side of a centerline, a paved shoulder, a gutter and curb, and a roadside grading. These components are defined independently as subassemblies. You can add any type of subassembly to make up a typical assembly, and then apply that assembly for a station range along an alignment.

Intersection objects have their own display style that controls the intersection object marker. Intersection components, such as offset alignments, curb return alignments, and corridors, also each have their own display styles. Before creating intersections, you should be familiar with creating and managing styles (page 53) and command settings (page 97). For more information on intersection styles, see Intersection Styles and Display (page 1425).

**Intersection Object Considerations**

It is important to understand the following before you begin creating intersections:

- Using the Create Intersection wizard, you can create an intersection object that contains up to four quadrants.

- If the default intersection assembly mapping and target assignments does not meet your design requirements, you can create the intersection using these components, and then edit the intersection regions manually to satisfy your design requirements.

- If you create an intersection, and then manually edit the intersection corridor regions, those changes will be overwritten if the intersection is regenerated (by clicking "Recreate Corridors" button on the Intersection tab) or by rebuilding the intersection, using the Rebuild Intersection command.

- When creating an intersection, targeting (both horizontally and vertically) is done automatically. If you want to change targets, you must manually modify the intersection corridor region(s) using the Corridor Properties dialog box.

- Typically, when an intersection object is created, the following objects are automatically added to the current drawing: intersection object, corridor(s), assemblies, subassemblies, offset alignments, and curb return alignments. However, assemblies, subassemblies, corridors, and profiles are only added if the appropriate options are selected.

# The Intersection Object

Intersection objects persist in an AutoCAD drawing as objects with the type AeccIntersection.

The intersection object allows you to create a 2D or 3D dynamic model of a T-shaped (3-way) intersection, or a four-way intersection.
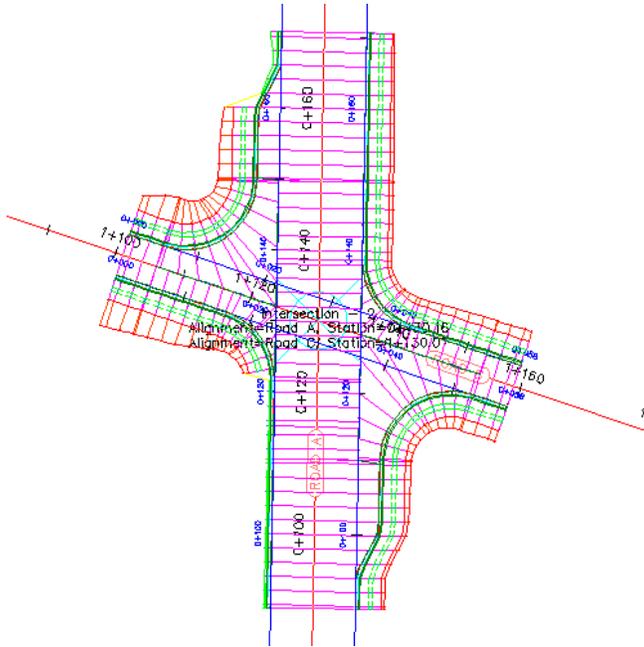


Before creating an intersection object, you must have two alignments that intersect. If you want to create a three-dimensional intersection model, you must also have profiles associated with the intersecting alignments. Once you have the desired prerequisites, then you can create the intersection object using the Create Intersection wizard.

When the intersection object is created, a variety of tasks can be performed automatically, including the following:

■ creates offset alignments for the roads included in the intersection

■ creates curb return alignments (curb returns) in the intersection area

■ creates profiles for the offset alignments and curb return alignments created when the intersection was created (Note: An exception to this is that if you set pre-defined static alignments as offsets, it is expected that you will specify profiles too. No dynamic offset profiles will be created.)

■ creates new corridor regions for the existing corridors in the intersection area, or creates new corridor objects in the intersection area

■ locks the profile elevations of the secondary road to the main (primary) road profile

Some of these tasks are optional. They depend on options selected on the wizard dialog boxes.

For overview information about AutoCAD Civil objects, see Understanding Objects and Styles (page 51).

## Intersection Styles and Display

Intersection object display style is controlled both by a specific intersection object style, and the styles and labels associated with the objects used to create the intersection, such as the intersecting alignments.

Use the intersection object style to control the appearance of the intersection object marker. For more information, see Intersection Style Dialog Box (page 1925).

Other components associated with an intersection, such as offset alignments, curb return alignments, and profiles, each have their own object style to control their appearance.

Similarly, when you have corridor objects associated with the intersection, various components within the corridor are controlled by their own display styles and labels. For more information, see Corridor Styles and Display (page 1329).

To set default styles for components that are created through the intersection creation command, use the default styles collection in command settings. For more information, see Changing Intersection-Related Default Style Settings (page 1438).

The following table lists the components of the intersection and where they derive their styles from.

| Intersection Component | Display Style | Notes |
| --- | --- | --- |
| Main and secondary road alignments | Alignment style | Defined in the existing alignment style |
| Offset alignments | Offset alignment style | Defined in the Create Intersection command settings |
| Curb returns | Curb return alignment style | Defined in the Create Intersection command settings |
| Offset profile | Curb return alignment style | Defined in the Create Intersection command settings |
| Curb return profile | Curb return alignment style | Defined in the Create Intersection command settings |

For more information about styles, see Object Styles (page 53).

# Intersection Collection (Prospector Tab)

Use the Intersection collection in the Prospector tree to access the intersection objects in a drawing. As intersection objects are created, they are displayed in the Intersections collection.

Expand the Intersections collection to view the names of the intersection objects included in the drawing. Expand an intersection object to display the Intersecting Alignments collection and the Intersection Quadrants collection. Click either of these collections to display a tabular list of the components in the Prospector list view. For more information, see The Toolspace Item View (page 82).

# Intersecting Alignments Collection

Use the Intersecting Alignments collection in the Prospector tree to access the alignments included in the intersection object.

Expand the Intersecting Alignments collection to view the names of the alignments included in the intersection. You can also click this collection to display the alignments in the Prospector list view. You can edit the alignment style or assign a different profile using this view. For more information, see The Toolspace Item View (page 82).

**Exploring Intersection Alignments in Prospector**

Another feature available from the Intersecting Alignments collection is the Explore In Prospector command.
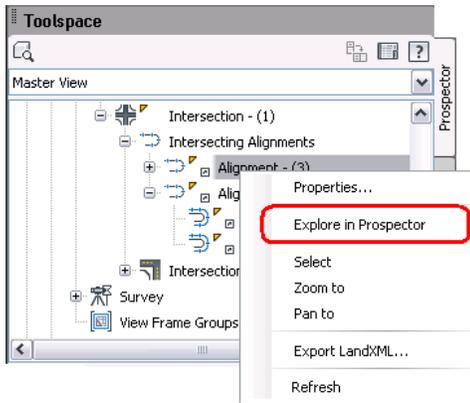
To use this command, right-click an alignment in the Intersecting Alignments collection, and select Explore In Prospector.

The same alignment is automatically displayed and selected in under the Alignments ➤ Centerline Alignments collection of the Prospector tree. Use this command to quickly find the location of an intersecting alignment in the Prospector tree.
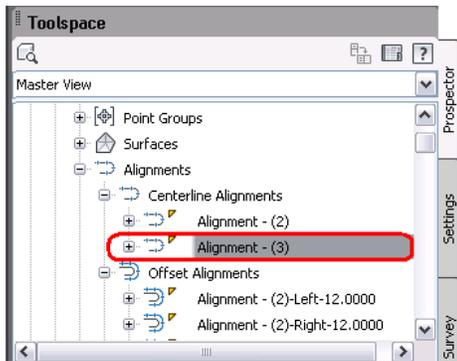


The Explore In Prospector command is available for centerline alignments, offset alignments, and curb return alignments, and is available from the Intersecting Alignments collection as well as from the Intersection Quadrants collection.

## Intersection Quadrants Collection

Use the Intersection Quadrants collection in the Prospector tree to access details about the quadrants in the intersection object.

Expand the Intersection Quadrants collection to view the names of the quadrant included in the intersection. This collection also displays the names of the curb return alignments for each quadrant. You can also use this collection to display the names of the incoming and outgoing lane centerlines in the Prospector list view. The list view also displays the value of the intersecting angle between the incoming and outgoing lane centerlines. For more information, see The Toolspace Item View (page 82).

Right-clicking an intersection quadrant name in Prospector lets you do the following:

| Item | Description |
| --- | --- |
| Edit Curb Return Parameters | Displays the Intersection Curb Return Parameters dialog box. You can change the quadrant name, or edit the curb return type and geometry (such as simple curve |

| Item | Description |
|---|---|
| | type). You can also add widening, or edit widening parameters. |
| Edit Curb Return Profile Rules | Displays the Intersection Curb Return Profile Parameters dialog box. You can change whether to extend the profile along the incoming or outgoing lanes, and also change the length by which to extend the profile. Editable parameters have a white background in the Value column, while read-only parameters have a shaded background. |
| Recreate Corridor Regions | Recreates corridor regions for the currently selected intersection. It is important to note that using this option removes any manual edits that may have been made to the intersection corridor. |
| Refresh | Refreshes the display of items displayed in Prospector. |

## Intersection Collection (Settings Tab)

Use the Intersection collection in the Settings tree to manage intersection object settings, including intersection object styles, label styles, and command settings.

An intersection object has its own display style that controls the appearance of the intersection object marker. Components included within the intersection object, such as alignments, profiles, corridors, and so on, each have their own display styles.

An intersection object style controls the intersection object marker appearance. For more information, see Intersection Style Dialog Box (page 1925).

Right-click the Intersection collection to do the following:

■ Edit the intersection feature settings. (page 1436)

■ Edit Intersection Label Style Defaults

■ Refresh the display of the settings tree.

Expand the Intersection collection to display and edit the intersection object style, label styles, and command settings that are available for the intersection feature.

## Creating Intersections

You can create an intersection using the Create Intersection wizard.
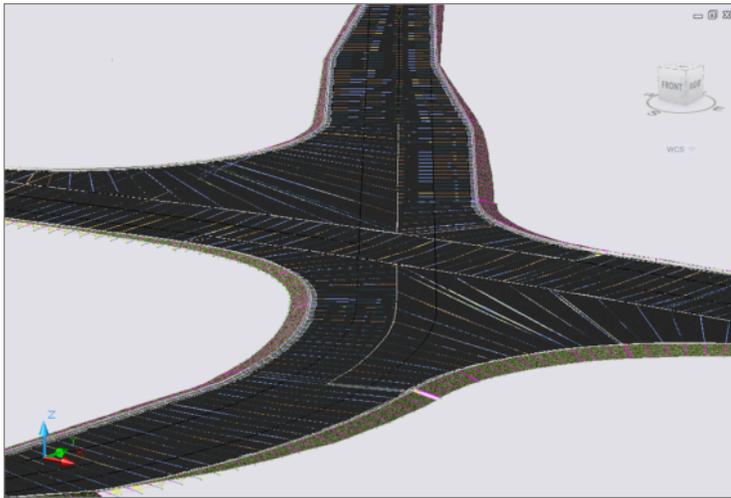
## Before You Begin

Before you can create a simple intersection, you must have at least two alignments that intersect each other only once in your drawing.

### Creating a Simple 2D Intersection

The simplest way to create a basic two-dimensional intersection object is to start with two alignments that intersect each other only once. You cannot use the Create Intersection wizard if the alignments do not intersect.

If your drawing has two intersecting alignments that meet this criteria, you can create a basic 2D intersection object using the Create Intersection wizard. Simply start the command, follow the prompts, and accept the default choices on the wizard dialog boxes.

### Creating a 3D Intersection with Corridor Objects, Alignments, and Profiles

To create a 3D intersection that includes a corridor, you will need to first create the road geometry (road centerline alignments and profiles) and existing ground surface in your drawing. Once these components exist in the drawing, you can proceed with using the Create Intersection wizard. The wizard prompts you to select the location in the drawing where the two road centerline alignments intersect.

You can start with the following data combinations:

■ two intersecting centerline alignments

■ two intersecting centerline alignments with one or more road edge offset alignments and or their profiles

You can have profiles defined already, or not. However, if profiles are not available, then features that are associated with 3D modeling will not be available on the wizard. For example, if you begin creating an intersection object with no profiles, then the following wizard options will not be available for selection: creating offset alignment and curb return profiles, as well as the ability to create a new corridor, or to add to an existing corridor in the intersection area.

With any of the scenarios just described, you can start the Create Intersection command, select the point where the centerline alignments intersect, and proceed with creating an intersection using the Create Intersection wizard.

If you need more information about creating the underlying objects required for creating 3D intersections, you can refer to the following table.

| For information about creating... | See... |
|---|---|
| Alignments | Creating Alignments (page 887) |
| Profiles | Creating Layout Profiles (page 1048) |
| Corridors | Creating Corridors (page 1330) |
| Assemblies | Creating Assemblies (page 1380) |
| Assembly Sets | Assembly Sets (page 1457) |
| Subassemblies | Creating Subassemblies (page 1396) |
| Surfaces | Creating Surfaces (page 601) |

# Setting Driving Direction

The Driving Direction option determines the direction in which curb returns are drawn when an intersection is created.

The options are Right Side of the Road and Left Side of the Road.

This setting should be set to **Left Side of the Road** when creating intersection objects for projects that have vehicles traveling on the left side of the road; for example, in countries such as the United Kingdom. In this case, curb return alignments in intersection objects are drawn starting on the left side of the outgoing road, and ending on the left side of the merging road.

When set to **Right Side of the Road**, curb return alignments in intersections are drawn starting from the right side of the outgoing road, and ending on the right side of the merging road.

**NOTE** You cannot change the "Driving Direction" setting on an intersection object that has already been created.

**To set driving direction**

1  In Toolspace, on the Settings tab, right-click the name of the drawing at the top of the Settings tree collection, and click Edit Drawing Settings.

2  Click the Ambient Settings tab, and expand the General collection.

3  Next to Driving Direction, click in the Value cell and select one of the two options (**Left Side of the Road** or **Right Side of the Road**.

## Quick Reference

Dialog Box

Ambient Settings Tab (Drawing Settings) (page 1856).

# Using the Create Intersection Wizard

To use the Create Intersection wizard, make sure you have the required data in your drawing before you begin.

As a minimum, you must start with a drawing that has two alignments that each have profiles and station data associated with them.

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**NOTE** If you are creating a T-shaped (3-way) intersection, you must ensure that the alignments snap together precisely and do not overlap. If there is any overlap between the two alignments, a 4-way intersection will be created.

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**To create an intersection using the Create Intersection wizard**

1 Do one of the following:

   ■ Click the Home tab ➤ Create Design panel ➤ Intersection ⊹ .

   ■ At the command line, enter CreateIntersection.

   You are prompted to pick an intersection point in the drawing.

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   **NOTE** If you wish to create a 3-way intersection, it is important that the alignments meet precisely and do not overlap or undershoot. If the two alignments overlap, a 4-way intersect is created.

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2 In the drawing, click the location in the drawing where two alignments intersect.

3 If you are creating a 4-way intersection, you will be prompted to select the alignment that will be identified as the primary road. If so, select the alignment by clicking the alignment in the drawing, or by pressing ENTER and selecting from the list of alignments in the drawing.

   If you are creating a T-shaped (3-way) intersection, then the alignment that passes through the intersection is automatically specified as the primary road (main road). In this situation, you are not prompted to specify which road is the primary road because this happens automatically. It is possible to change the road priority only during intersection object creation (on the Geometry Details page of the Create Intersection wizard). For more information, see Changing Road Priority (page 1433).

4 On the General page of the Create Intersection wizard, enter a name for the intersection object, or accept the default. You can also enter an optional description, and change defaults for marker styles, layer, and labels.

5 For the Intersection Corridor Type, select one of the following to specify the corridor grade options through the intersection area:

   ■ Primary Road Crown Maintained: When this options is selected, the primary road crown is maintained through the intersection.

   ■ All Crowns Maintained: When this options is selected, the crowns of both roads included in the intersection are maintained through the intersection.

   An illustration at the bottom of the wizard dialog box provides a conceptual graphic of the grade type selected.

6 Click Next to proceed to the next page of the wizard.

7 On the Geometry Details page, you may accept or change the defaults. For example, if desired you can perform the tasks listed in the following table:

| To perform this option... | do this... |
| --- | --- |
| Change the alignment priority | Select an alignment in the list and click the arrows. For more information, see Changing Road Priority (page 1433). |
| Select different profiles for the alignments | Select an alignment in the list and click in the Profile cell to select a profile from the |

| To perform this option... | do this... |
| --- | --- |
| | drop-down list of available profiles in the current drawing. |
| Choose to create or not create offset alignments and curb returns for the intersection | Place or remove the check mark from the appropriate box on this wizard dialog box. |
| Change parameters for the offset alignments, curb returns, lane slopes, curb return profiles | Click the appropriate Parameters button on this wizard dialog box. |

8   By default, new offset alignment objects will be created. However, if you want to specify an existing offset alignment from the drawing, click the Offset Parameters button, set Use An Existing Alignment to Yes, and select the alignment by clicking ⊡ next to the Alignment Name property.

9   When you are finished configuring the Geometry Details page, click Next to proceed to the next page of the wizard.

10  On the Corridor Regions page, you may accept or change the defaults. For example, if desired you can perform the tasks listed in the following table:

| To perform this option... | do this... |
| --- | --- |
| Create or not create a new corridor in the intersection area | Add or remove the check mark next to this option. For more information, see Creating Corridors in the Intersection Area (page 1435). |
| Add to an existing corridor | Select this option. If you do not have corridor objects in the drawing, this option is not available. For more information, see Creating Corridors in the Intersection Area (page 1435). |
| Select a surface to daylight | If a surface is available in the current drawing, you can select a surface to daylight. However, you cannot target a surface that is an output of another corridor. |
| Select an assembly set to import | Click Browse, select the assembly set, then ensure the assembly set is selected using the drop-down arrow under Select Assembly Set To Import. For more information, see Assembly Sets (page 1457). |
| Select different assemblies | If the desired assembly objects exist in the current drawing, you can select different assemblies by c licking ⊡ in the Assembly To Apply column. Otherwise, accept the default assemblies specified. |

11  Click the items listed under Corridor Region Section Type, and notice that the conceptual graphics update with each selection. The location where the assembly will be applied along the intersection is highlighted in the conceptual graphic.

**12** Click Create Intersection. The intersection is created.

### Quick Reference

Ribbon

Home tab ➤ Create Design panel ➤ Intersection 🔳

Command Line

CreateIntersection

Dialog Box

Create Intersection Wizard (page 1927)

## Changing Road Priority

When you create an intersection object, a Priority value is assigned to the main (primary) and secondary (side) road alignments .

The following table lists the value of the Priority property:

| Road (alignment) | Priority |
| --- | --- |
| Main (primary) road alignment | 1 |
| Secondary road alignment | 2 |

If you are creating a T-shaped (3-way) intersection, then the alignment that passes through the intersection is automatically specified as the primary road (main road). In this situation, the Priority is automatically set to a value of 1 for that alignment, and you are not prompted to specify which road is the primary road because this happens automatically.

You cannot change the priority of a road in an intersection after it has been created. However, if you want to change the priority of the alignments *during* the intersection creation, you can do so from the Geometry Details page of the Create Intersection wizard.

It is also important to note that typically, the secondary road (or lower priority) profile is automatically locked at the intersection location to the elevation of the main road (higher priority) main road. The profile of the main road is not modified.

**To change road (alignment) priority during intersection creation**

**1** Click the Home tab ➤ Create Design panel ➤ 🔳 Intersection.

**2** Advance to the Geometry Details page of the Create Intersection wizard.

**3** At the top of this dialog box, in the Intersecting Alignments section, select the road (alignment) you want to change by clicking the name of the alignment in the list.

**4** Click the up or down arrow to change the road priority.

> **NOTE** Once the intersection object is created, you cannot change the value of the road priority for an alignment.

**5** Continue creating the intersection. For more information, see Using the Create Intersection Wizard (page 1430).

**Quick Reference**

Dialog Box

Geometry Details Page (Create Intersection Wizard) (page 1929)

# Intersection Corridor Type

In AutoCAD Civil, the Intersection Corridor Type property identifies the type of grading that will be applied to roads in an intersection object.

There are two types of intersection grading options that are available with AutoCAD Civil intersection objects: Primary Road Crown Maintained and All Crowns Maintained. You must specify this type when you create the intersection, on the General page of the Create Intersection wizard.

For existing intersections, the intersection corridor type is displayed on the Intersection Details tab of the Intersection Properties dialog box, in the Intersection Quadrants section. You can set the creation default for this option on the Edit Command Settings - CreateIntersection dialog box (page 1919).

Once the intersection is created, you cannot change the value of the Intersection Corridor Type property. If you do want to change this option after the intersection object is created, you must delete the intersection object, and create a new one with the desired Intersection Corridor Type.

### Primary Road Crown Maintained

In this type of intersection, the crown of the primary road is maintained, while the crown (profile) of the secondary road is adjusted to match the edge of the primary road and the intersection point. The crown (profile and edges) of the primary road are not affected.

The following conceptual graphic is displayed on the wizard dialog when you select this option:
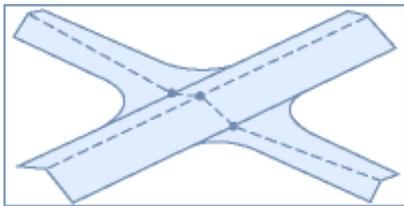


This represents how the grading occurs for the intersection object.

### All Crowns Maintained

In this type of intersection, the profile of the side road is adjusted to match the main road elevation at the intersection point. The main road profile is not affected. The edges are blended together along the curb returns (via the curb return profiles).

The following conceptual graphic is displayed on the wizard dialog when you select this option:
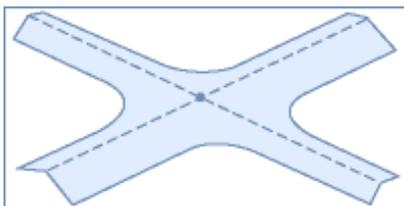


**NOTE** Once an intersection object is created, you cannot change the value of the intersection Corridor Type property.

# Creating Corridors in the Intersection Area

You can choose to create corridor objects within intersections, or add to existing corridors. You can also choose not to have corridor objects in the intersection.

In order to create the intersection corridor, the following must have already been done:

- Side road profile matching
- Creation or selection of offsets and profiles
- Creation of curb returns and profiles

If the above items have not been met, then the create corridor options on the Create Intersection wizard dialog box are not available.

The following table describes the options that are provided through the Create Intersection wizard dialog box:

| Option | Description |
| --- | --- |
| Create a New Corridor | Specifying this option creates a new corridor object in the intersection area. |
| Add to an Existing Corridor | Specifying this option lets you select an existing corridor object in the drawing to add to the intersection area. |
| No Corridors in the Intersection Area | When neither of the previouos two options are selected, no corridor object are created in the intersection area. |

You can specify these options during intersection object creation, using the Create Intersection wizard, or you can choose to edit these options after the intersection object has been created.

For example, you can select an existing intersection object, and click Recreate Corridor Regions from the shortcut menu. This displays the Intersection Corridor Regions dialog box which lets you edit corridor regions on the intersection. From this dialog box, you can change the options selected for corridor object creation. After making changes on this dialog box, you need to recreate the corridor regions for the intersection area.

**To create a new corridor in the intersection during creation**

This procedure describes how to create a new corridor in an intersection area during intersection object creation.

1. Start the Create Intersection wizard by doing one of the following:
   - Click the Home tab ➤ Create Design panel ➤ Intersection .
   - At the command line, enter CreateIntersection.

   You are prompted to pick an intersection point in the drawing.

2. Configure the first two pages of the wizard as desired, and advance to the Corridor Regions page. For more information, see Using the Create Intersection Wizard (page 1430).

3. On the Corridor Regions page, place a check mark in the Create Corridors in the Intersection Area check box.

4. Select the Create a New Corridor option.

**5** Proceed with configuring the rest of the wizard options.

**To add to an existing corridor in the intersection during creation**

This procedure describes how to add to an existing corridor in an intersection area during intersection object creation.

**1** Start the Create Intersection wizard by doing one of the following:

■ Click the Home tab ➤ Create Design panel ➤ Intersection 🔧.

■ At the command line, enter CreateIntersection.

You are prompted to pick an intersection point in the drawing.

**2** Configure the first two pages of the wizard as desired, and advance to the Corridor Regions page. For more information, see Using the Create Intersection Wizard (page 1430).

**3** On the Corridor Regions page, place a check mark in the Create Corridors in the Intersection Area check box.

**4** Select the Add to an Existing Corridor option.

**5** Select an existing corridor using the drop-down list or by picking from the drawing.

**NOTE** If you do not have corridor objects in the drawing, these options is not available.

**6** Proceed with configuring the rest of the wizard options.

**To create a new, or to add to an existing corridor in an intersection**

This procedure describes how to edit an existing intersection object to either add a new corridor object to it, or to add an existing corridor object to it.

**1** Select an intersection object, and then click Recreate Corridor Regions.

The Intersection Corridor Regions dialog box is displayed.

**2** On the Intersection Corridor Regions dialog box, select one of the following: Create a New Corridor or Add to an Existing Corridor.

**3** If you selected, Add to an Existing Corridor, select an existing corridor using the drop-down list or by picking from the drawing.

**NOTE** If you do not have corridor objects in the drawing, this option is not available.

**4** Configure the rest of the choices on this dialog box. For example, you may want to select a different assembly set, or select different assemblies. For more information, see Intersection Corridor Regions Dialog Box (page 1935).

**5** To apply the changes, click the Recreate button on this dialog box.

**NOTE** Because this command deletes and recreates the corridors in the intersection, if you made any changes to the corridors in the intersection object, those changes will be overwritten (discarded) when the new corridors are created. For more information, see Recreating Corridor Regions in Intersections (page 1454).

## Editing Intersection Settings

Use intersection settings to specify the default behavior for intersection-related commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Toolspace Settings tree. You can control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

In the Settings tree, use the Intersection collection's shortcut menu to establish defaults for all intersection-related commands. You can change intersection-specific settings at this level, or override the drawing ambient settings.

Use the Commands collection in the Intersection collection to change intersection settings for a specific command.

**NOTE** Overrides to the drawing ambient settings at the Intersection collection level and the Commands collection level affect only the specified (and subordinate) levels. The drawing level settings are not changed.

The topics in this section describe only those settings that affect intersection-related commands. They do not apply to the drawing ambient settings that you can change at the Intersection collection level. For more information about the drawing ambient settings, see Specifying Ambient Settings (page 68).

# Changing Intersection Creation Settings

Use the intersection creation and naming settings in the Edit Feature Settings - Intersections dialog box to change intersection-related settings before you create intersections.

For example, before creating an intersection, you may want to specify the default frequency for inserting assemblies along tangents, curves, or spirals. You may also want to set default parameters for curb returns, widening, or curb return profiles.

If a closed lock icon 🔒 is displayed in the Lock column for a property, the property is locked at a higher level of the Settings tree. It cannot be changed at this level.

If you change a property value, a check mark ✔ is displayed in the Override column. This override is also noted in the property table for related objects higher up the Settings tree, where an arrow is displayed in the Child Override column.

**To change intersection creation settings**

1  Do one of the following:

   ■ **To edit settings for all intersection-related commands**: In Toolspace, on the Settings tab, right-click the Intersection collection. Click Edit Feature Settings.

   ■ **To edit settings for a specific command**: In Toolspace, on the Settings tab, expand the Intersection collection. Expand the Commands collection. Right-click the name of the command and click Edit Command Settings. For example, right-click the CreateIntersection command and click Edit Command Settings.

   TheEdit Feature Settings - Intersection dialog box (page 1919) is displayed.

2  To specify defaults for intersection creation, right-click the CreateIntersection command and click Edit Command Settings. For example, to edit default frequencies for inserting assemblies along tangents, curves, and spirals, expand Assembly Insertion Defaults. For more information on intersection defaults, see the Edit Feature Settings - Intersection (page 1919) dialog box.

3  To edit the default naming format for intersection data, expand the Default Name Format property group. Edit the various name template settings.

4  Click Apply.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: Right-click Intersection collection ➤ Edit Feature Settings
OR
Settings tab: Intersection ➤ Commands ➤ Right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Intersection (page 1919)

# Changing Intersection-Related Default Style Settings

Use the intersection settings to specify default styles and labels for intersection creation and for certain components that are create during intersection creation.

In addition to selecting a default style for the intersection object marker, you can also select default styles for the following objects that can be created when an intersection object is created:

- Intersection label

- Offset alignment

- Curb return alignment

- Offset alignment profile

- Curb return alignment profile
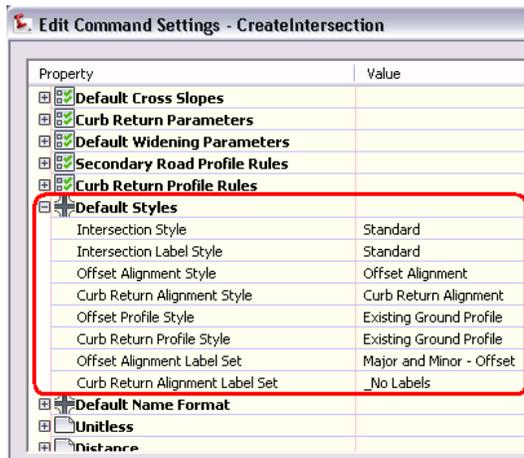


You can also create label sets for offset alignments and for curb return alignments.

The default styles specified for these components are used to establish the default styles that are used when these items are created using the CreateIntersection command.

**To change the intersection-related style and label settings**

1  In Toolspace, on the Settings tab, right-click the Intersection collection. Click Edit Feature Settings to open the Edit Feature Settings - Intersection dialog box (page 1919).

2  Expand the Default Styles property group.

**3** To specify a default style for intersection output objects, such as offset alignments, curb returns, and profiles generated from intersections, click the corresponding field. Click ⋯ to open the Select Style dialog box.

**4** To specify a default intersection style, click Intersection Style. Click ⋯ to open the Select Style dialog box.

**5** To specify a default style for a label set, click the corresponding field. Click ⋯ to open the Select Label Set dialog box.

**6** Click Apply.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click Intersection collection ➤ Edit Feature Settings

Dialog Box

Edit Feature Settings - Intersection (page 1919)

# Editing Intersections

There are a variety of ways that you can edit intersections.

You can edit intersections using commands available from the ribbon, from right-click shortcut menus, or by editing the objects directly in the drawing using grips.

The following lists describe some of the ways that you may want or need to edit an intersection object after it has been created.

### Changes Managed by the Intersection Object

The following types of edits can be performed by making changes directly to components that are controlled by the intersection object:

■ Change the parametric values of the intersection object definition, such as nominal offsets/ cross slopes, curb return profile rules, and corridor regions

■ Change edge offsets from parametric (nominal offset) values to predefined alignments

■ Change edge offset profiles from parametric (nominal cross slopes) to predefined profiles

■ Change (intersection) rule-based dynamic offset alignments and dynamic profiles to static and then make edits to them

■ Change association of assemblies to one or more of "dynamic corridor regions"

■ Change the frequency that the assemblies are applied to the dynamic corridor regions

Typically you will need to update corridor regions after making these types of edits to intersection objects.

When an update is necessary, an out-of-date icon ⚠ is displayed in Prospector for the corridors in the intersection. If you have Update Regions Automatically turned on for the intersection object, then the corridor regions are updated automatically after these types of edits. For more information, see Updating Corridor Regions in Intersections (page 1452).

**Changes Not Managed by the Intersection Object**

While you can make the above types of changes by manipulating components that are controlled by the intersection object, it is also possible to make other types of changes that are not directly managed by the intersection object. For example, you can make the following types of changes that are not managed by the intersection object:

■ Change the intersection area corridor region extents and or add more regions within the intersection extents

■ Change the corridor targets in the intersection for the edge offsets and profiles

■ Edit corridor parameters in the intersection area regions using the Corridor Section View Editor

Because these last three types of changes do not modify the intersection object definition, it is important to understand that if you do make these types of changes, they could be overwritten or ignored if you update the corridor regions in the intersection, or if you recreate the intersection.

**NOTE** When you edit the horizontal or vertical geometry of the intersection object, the road that is specified as the secondary road is automatically adjusted to accommodate the horizontal and vertical position of the road that is specified as the primary road.

The sections that follow describe how to make various types of edits to intersection objects.

# Editing Basic Intersection Information

After creating an intersection, you can use the Prospector tree to edit basic intersection object information, such as object name and description.

**To edit basic intersection information**

1 In Toolspace, on the Prospector tab, expand the  Intersection collection. Right-click the intersection. Click Properties.

2 In the Intersection Properties dialog box, click the Information tab (page 1924).

3 To change the name of the intersection, enter a new name in the Name field.

4 To change the description of the intersection, enter a new description in the Description field.

5 To change the style of the intersection, click the Object Style list. You can also use the standard buttons to create, edit, or select a style.

6 Click Apply to make the changes, or click OK to make the changes and close the Intersection Properties dialog box.

## Quick Reference

Toolspace Shortcut Menu

  Prospector tab: Intersections ➤ <intersection name> ➤ Properties ➤ Information tab

Object Shortcut Menu

  Intersection Properties

Dialog Box

  Intersection Properties - Information tab (page 1924)

# Editing Intersection Properties

You can change the intersection object name, style, alignment profiles, and view details about the alignments and quadrants in the current intersection using the Intersection Properties dialog box.

The Intersection Properties dialog box has the following tabs:

■ **Information**. Specifies intersection object name, description, and style.

■ **Intersection Details**. Displays information about the alignments and the quadrants included in the intersection. This tab displays the names of the alignments, the station of the point of intersection, and profile associated with each alignment. If multiple profiles are available, you can change the profile associated with the alignment by clicking in the Profile cell and selecting a new profile from the drop-down list. The Intersection Quadrants section displays the names of the alignments included in each quadrant, and identifies incoming and outgoing alignments, as well as the value for the intersection angle between the two alignments. This is for information only and cannot be edited in the properties dialog.

**See also:**

■ Intersection Properties dialog box (page 1924)

**To change intersection object name, description, or the intersection marker style**

1 Right-click an intersection in the drawing. Click Intersection Properties.

> **NOTE** The tab most recently opened in the Intersection Properties dialog box is displayed by default.

2 In the Intersection Properties dialog box, click the Information tab (page 1924).

3 Under Name, enter a name for the intersection object.

4 Under Description, enter an optional description for the intersection.

5 In the Object Style list, select an existing intersection style. Note that this only affects the intersection marker style.

6 Click the Style Detail button to open the Style Detail dialog box. Preview the intersection marker style.

**To change the profile associated with an alignment**

1 Right-click an intersection in the drawing. Click Intersection Properties.

> **NOTE** The tab most recently opened in the Intersection Properties dialog box is displayed by default.

2 In the Intersection Properties dialog box, click the Intersection Details tab (page 1925).

3 In the Alignment Details section, click a profile name in the Profile column. Select a profile from the drop-down list. If there are not multiple profiles in the drawing, no other profiles will be displayed.

4 Click Apply to apply the new profile.

5 After changing the profile, you may also need to rebuild the corridor, recreate corridor regions in the intersection, or rebuild the intersection. For more information see Rebuilding a Corridor (page 1360), Recreating Corridor Regions in Intersections (page 1454), or Updating Corridor Regions in Intersections (page 1452).

### Quick Reference

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Properties

Object Shortcut Menu

Right-click <intersection object> ➤ Intersection Properties

Command Line

EditIntersectionProperties

Dialog Box

Intersection Properties Dialog Box (page 1924).

## Editing Intersection Styles

Use styles to control the visual display of each intersection component.

The intersection style controls the appearance of the intersection object marker. You can change the marker, as well as display parameters such as visibility, layer, and color used in plan and model views.

Access intersection styles using these methods:

| If you want to access intersection styles... | Then... |
| --- | --- |
| From the ribbon | Select an intersection ➤ Intersection tab ➤ Modify panel ➤ Intersection Properties drop-down ➤ Edit Intersection Style |
| In the drawing | Right-click an intersection ➤ Edit Intersection Style. |
| On the Settings tab in Toolspace | Right-click a style in the Intersection Styles folder ➤ Edit. |

**NOTE** When you click New in a collection folder, the new style you create is based on installation defaults, not on the existing styles in the collection. To create a style from an existing style, right-click the style. Click Copy and save the style with a new name.

Use the tabs in the Intersection Style dialog box to set style information. For more information, see Intersection Style Dialog Box (page 1925).

The following intersection components derive their styles from their own object styles: primary and secondary road alignments, offset alignments, curb return alignments, profiles, corridors.
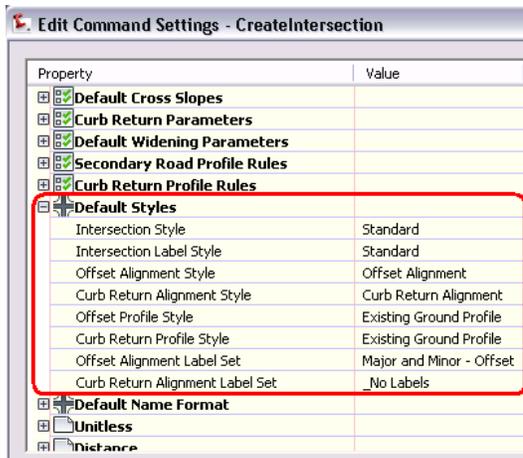
Use the CreateIntersection command settings to set the default styles that will be used when creating some of these components.

## Intersection Editing Visual Cues

Visual cues help you edit intersection components quickly.

Many of the dialog boxes that allow you to create or edit intersection object components have visual behavior built in to help you quickly identify the area of the intersection object being edited.

For example, editing dialog boxes, such as the Intersection Offset Parameters dialog box and the Intersection Curb Return Parameters dialog box, contain the following behaviors:

**Object Highlighting in Drawing:** When you select some items on an intersection editing dialog box, the affected object is highlighted in the drawing. For example, when you select an offset on the Intersection Offset Parameters dialog box, that offset alignment is highlighted in the drawing. This behavior is displayed in the following illustration.
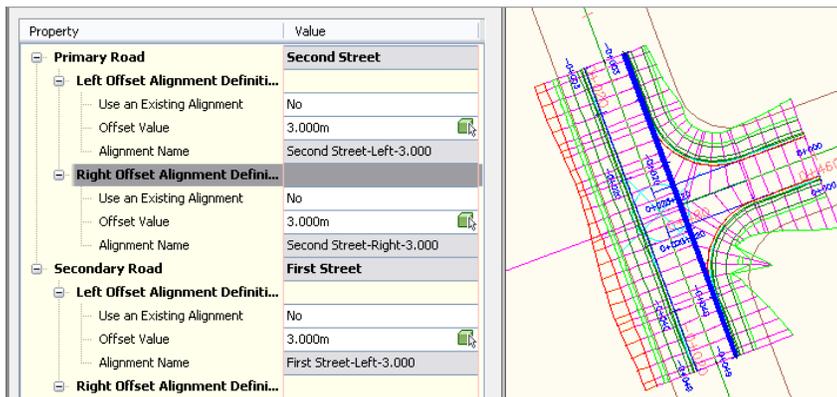


The right side offset alignment named First Street is selected in the Intersection Offset parameters dialog box, shown on the left. In the drawing, that offset alignment is highlighted.

Similarly, if you are editing intersection curb return parameters using the Intersection Curb Return Parameters dialog box, each time you select a quadrant or curb return in this dialog box, the selected quadrant or curb return is highlighted in the drawing.

The following illustration shows temporary arrows along the alignments that indicate the direction of incoming and outgoing traffic. These arrows are displayed when you are specifying intersection curb return parameters.
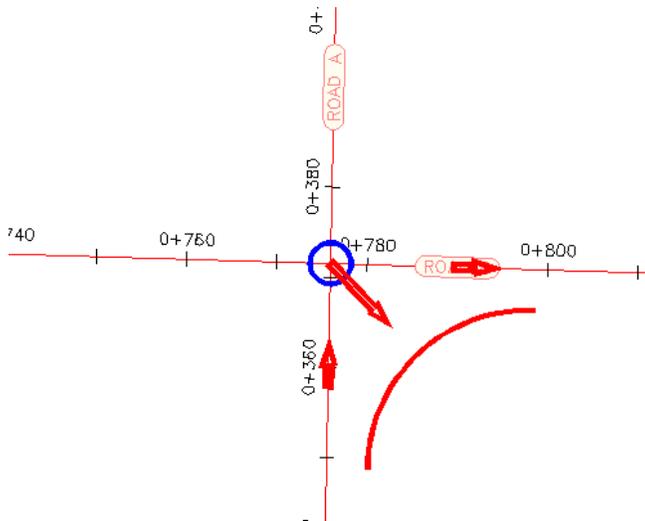
**Conceptual Graphics on Dialog Boxes:** Each time you select a property or item to edit on intersection editing dialog boxes, the conceptual graphic is displayed at the bottom of the editing dialog box updates to indicate the location along the intersection affected by the edit.

For example, when you select the "Primary Road Full Section" corridor region on an intersection editing dialog box, the following conceptual graphic is displayed, indicating the location of this region along the corridor.
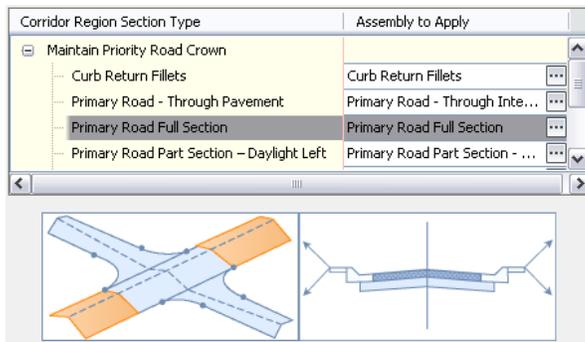


Similarly, when you select the "Primary Road Full Section" corridor region
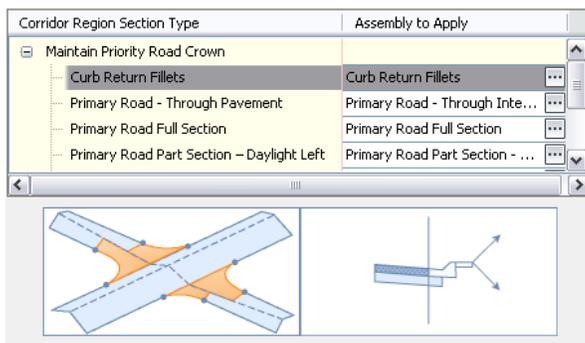


This behavior also exists on the Create Intersection wizard dialog boxes.

**Shading of Non-editable Properties:** In editing dialog boxes, properties that are not editable are displayed with a gray (shaded) background color. Properties that are editable are displayed with a white (not shaded) background color.

# Editing Offsets in Intersections

There are several ways you can edit offset alignments in an intersection area.

You may need to edit offset alignments after an intersection is created. You can change the offset value, or select a different alignment to use as the offset.

For example, you can use the Edit Offsets command on an existing intersection object to change the parametric value of the intersection, or you can select a different offset alignment, using one from the current drawing. You can also grip edit the dynamic offset alignment after it is created.

It is also possible to edit offsets for incoming and outgoing roads by editing the corridor parameters, although this is not typically the preferred method. For example, the offsets can be changed by overriding offsets at one or more stations, or by overriding target parameters for incoming and outgoing lanes.

**Visual Cues**: When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location of the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**To edit offset alignments using the Edit Offsets command**

This procedure describes how to widen a road by editing the offset alignment definition on an existing intersection object.

1  Do one of the following:

   ■  In the drawing, click the intersection object. On the ribbon, click Edit Offsets ⊞ on the Modify panel of the Intersection contextual tab.

   ■  In the drawing, right-click the intersection object, and then select Edit Offset Parameters from the object shortcut menu.

   ■  In Prospector, expand the Intersections collection, right-click the intersection object, and then select Edit Offset Parameters from the Toolspace shortcut menu.

2  On the Intersection Offset Parameters dialog box, expand the collection for the alignment you want to edit.

   Notice that when the Use An Existing Alignment property is set to No, the Alignment Name property is read only. Notice also how the Offset Value property is set, and that it is currently editable.

3  Set the Use An Existing Alignment property to Yes, and notice that the Alignment Name property becomes editable.

4  Click ⊡ to select another alignment from the current drawing.

   Notice that the Offset Value is set to *Varies* if the alignment is not a dynamic offset alignment based on a centerline. Otherwise, the Offset Value specifies the offset value of the attached dynamic alignment. This control is disabled (read only) in both cases.

   **NOTE**  If you select another alignment in the Alignment Name field, the original offset alignment that was created during intersection object creation is not deleted.

**To specify offset alignment width during intersection creation**

This procedure describes how to widen a road by specifying a different offset alignment using the Create Intersection wizard.

1 Do one of the following:

■ Click Home tab ➤ Create Design panel ➤ Intersection 🔧 .

■ At the command line, enter CreateIntersection.

2 Pick the intersection point in the drawing.

3 In the wizard, click Next to proceed to the Geometry Details page.

4 Click Offset Parameters.

5 On the Intersection Offset Parameters dialog box, expand the collection for the alignment you want to edit.

6 Set the Use An Existing Alignment property to Yes, and notice that the Offset Alignment Name Format property is replaced with the Alignment Name property.

7 In the Value field for the Alignment Name property, click ⬚ to select an offset alignment from the current drawing.

Notice that the Offset Value is set to *Varies* if the alignment is not a dynamic offset alignment based on a centerline. Otherwise, the Offset Value specifies the offset value of the attached dynamic alignment.

---

**NOTE** When you assign an offset value to an existing alignment, either dynamic or static, the Intersection object does not remember the original dynamic object associated with it.

---

8 Click OK on the Intersection Offset Parameters dialog box, and proceed with using the wizard to create the intersection object. For more information, see Using the Create Intersection Wizard (page 1430).

**To edit offset alignments using grips**

This procedure describes how to widen a road by grip editing the offset alignment that is controlling an intersection.

1 In the drawing, select the desired offset alignment in the intersection object.

2 On the offset alignment, click and drag the Change Offset (triangle grip) to specify the new offset value. Notice that the intersection object and the associated curb returns update.

## Quick Reference

Ribbon

Click Intersection tab ➤ Modify panel ➤ Edit Offsets 🔧

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Edit Offsets

Object Shortcut Menu

Right-click <intersection object> ➤ Edit Offsets

Command Line

EditIxOffsets

Dialog Box

Intersection Offset Parameters Dialog Box (page 1931)

## Editing Curb Returns in Intersections

You can edit curb returns in an intersection.

You may need to edit curb returns after an intersection is created. For example, you can change the curb return type or dimension, or edit widening or transition details.

**Visual Cues**: When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location of the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**To edit curb returns in an intersection**

1   Do one of the following:

   ■ In the drawing, click the intersection object. On the ribbon, click Edit Curb Returns ⊞ on the Modify panel of the Intersection contextual tab.

   ■ In the drawing, right-click the intersection object, and then select Edit Curb Return Parameters from the object shortcut menu.

   ■ In Prospector, expand the Intersections collection, right-click the intersection object, and then select Edit Curb Return Parameters from the Toolspace shortcut menu.

2   Use the Intersection Curb Return Parameters dialog box to edit the following parameters associated with this component.

   ■ Intersection Quadrant: You can specify which quadrant of the intersection to edit.

   ■ Widen Turn Lane For Incoming\Outgoing Road: These options automatically add widening to the incoming or outgoing road in the intersection when the intersection is created.

   ■ Intersection Quadrant Name: Displays the name of the intersection quadrant. You can change the name here, if desired.

   ■ Incoming Road Centerline Name: Displays the name of the centerline alignment for the incoming road.

   ■ Outgoing Road Centerline Name: Displays the name of the centerline alignment for the outgoing road.

   ■ Intersection Quadrant Angle: Specifies the angle in degrees that is created by the two alignments that intersect, forming this intersection quadrant.

   ■ Curb Return Type: Specifies the type of curb return, either chamfer, circular fillet, or 3-centered arc.

   ■ Radius: When the curb return type is circular fillet or 3-centered arc, you can specify radius parameters for the curb return.

   ■ Copy These To All Quadrants: On this dialog box, you can right-click the Curb Return Parameters node in the Property column, and select Copy These To All Quadrants. When this option is selected, the curb return parameters currently selected are automatically applied to all quadrants in the intersection.

   ■ Widening At Incoming\Outgoing Lane: Specifies details for the widening at the incoming and outgoing lanes in the intersection.

- Transition Details: Specifies the transition details for the incoming or outgoing lanes in the intersection.

- Widening Details: Specifies the widening details for the incoming or outgoing lanes in the intersection.

### Quick Reference

Ribbon

Click Intersection tab ➤ Modify panel ➤ Edit Curb Returns

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Edit Curb Returns

Object Shortcut Menu

Right-click <intersection object> ➤ Edit Curb Returns

Command Line

EditIxOffsets

Dialog Box

Intersection Curb Return Parameters Dialog Box (page 1932)

## Editing Side Road Profiles for Intersections

You can edit the rules to adjust the profile of the side road in an intersection.

You may need to edit the profile for a side road after an intersection is created. For example, you can apply or not apply grade rules, or change the maximum grade.

This helps set reasonable grades coming in and out of the intersection on the side road.

**Visual Cues**: When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location of the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**To edit a side road profile in an intersection**

1 Do one of the following:

- In the drawing, click the intersection object. On the ribbon, click Side Road Profile  on the Modify panel of the Intersection contextual tab.

- In the drawing, right-click the intersection object, and then select Adjust Side Road Profile from the object shortcut menu.

- In Prospector, expand the Intersections collection, right-click the intersection object, and then select Adjust Side Road Profile from the Toolspace shortcut menu.

---

**NOTE** If one or more of the dynamic PVIs created by the intersection are unlocked, a task dialog is displayed prompting you to either relock the PVIs to match the primary road, or to continue the command without relocking the PVIs. If they do not match the primary road elevation, the corridor top surface in the curb return fillet areas will be uneven. Choose to either relock the PVIs, if possible, or to continue the command without relocking the PVIs.

---

**2** Use the Secondary Road Profile Rules dialog box to edit the following parameters associated with this component.

■ Apply Grade Rules: Specifies whether to apply grade rules for this secondary road profile.

■ Maximum Grade: Specifies the maximum grade that can be used for the tangents leading in to or leaving the intersection along the secondary road.

■ Maximum Grade Change: Specifies the maximum grade change between the intersection area and the tangent leading to the intersection along the secondary road.

■ Distance Rule to Adjust the Grade: Specifies to use a user-defined length or to the extents of the intersection, as lengths, for incoming and outgoing grades along the secondary roads.

■ Distance Value: When using the Distance Rule to Adjust the Grade option, this specifies the distance rule value used for the grade used for the secondary road in the intersection.

## Quick Reference

Ribbon

Click Intersection tab ➤ Modify panel ➤ Side Road Profile 🔧

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Adjust Side Road Profile

Object Shortcut Menu

Right-click <intersection object> ➤ Adjust Side Road Profile

Command Line

EditIxSideRoadProfile

Dialog Box

Secondary Road Profile Rules Dialog Box (page 1933)

## Editing Lane Slopes for Intersections

You can edit lane slopes in an intersection.

You may need to edit lane slopes after an intersection is created. For example, you can change the cross slope from centerline, or select a different profile to use.

It is importnat to note that if you manually specify a lane slope by selecting a profile for the left edge (for example), then the lane slope is determined by that profile.

**Visual Cues**: When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location of the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**To edit lane slopes in an intersection**

**1** Do one of the following:

■ In the drawing, click the intersection object. On the ribbon, click Lane Slopes 🔁 on the Modify panel of the Intersection contextual tab.

■ In the drawing, right-click the intersection object, and then select Edit Lane Slope Parameters from the object shortcut menu.

- In Prospector, expand the Intersections collection, right-click the intersection object, and then select Edit Lane Slope Parameters from the Toolspace shortcut menu.

2 Use the Intersection Lane Slope Parameters dialog box to edit the following parameters associated with this component.

- Left or Right Edge Profile Definition: Specifies the parameters defining the left and right edge of the slope

- Use an Existing Profile: When Yes is selected, you can select a profile from the available profiles in the current drawing. When No is selected, this indicates that the profile was created during intersection creation.

- Cross Fall From Centerline: Specifies the slope of the grade from the centerline.

- Profile Name: Displays the name of the profile associated with this edge profile definition. When Use An Existing Profile is set to Yes, you can select a new profile from available profiles in the current drawing.

### Quick Reference

Ribbon

Click Intersection tab ➤ Modify panel ➤ Lane Slopes

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Edit Lane Slope Parameters

Object Shortcut Menu

Right-click <intersection object> ➤ Edit Lane Slope Parameters

Command Line

EditIntersectionLaneSlopes

Dialog Box

Intersection Lane Slope Parameters Dialog Box (page 1934)

## Editing Curb Return Profiles for Intersections

You can edit curb return profiles in an intersection.

You may need to edit curb return profiles after an intersection is created. For example, you can extend or not extend the profile along the incoming or outgoing lanes.

**Visual Cues**: When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location of the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**To edit curb return profiles in an intersection**

1 Do one of the following:

- In the drawing, click the intersection object. On the ribbon, click Curb Return Profiles on the Modify panel of the Intersection contextual tab.

- In the drawing, right-click the intersection object, and then select Edit Curb Return Parameters from the object shortcut menu.

- In Prospector, expand the Intersections collection, right-click the intersection object, and then select Edit Curb Return Parameters from the Toolspace shortcut menu.

2 Use the Intersection Curb Return Parameters dialog box to edit the following parameters associated with this component.

- **Extend Profile Along Incoming (or Outgoing) Lane**: Specifies whether to extend the curb return profile along the incoming (or outgoing) lane of the intersection.

- **Length To Extend Along Incoming (or Outgoing) Lane**: When **Extend Profile Along Incoming (or Outgoing) Lane** is set to Yes, this option lets you specify the length to extend the curb return profile along the incoming (or outgoing) lane of the intersection.

## Quick Reference

Ribbon

Click Intersection tab ➤ Modify panel ➤ Curb Return Profiles 

Toolspace Shortcut Menu

Prospector tab: Right-click <intersection item > ➤ Edit Curb Return Profiles

Object Shortcut Menu

Right-click <intersection object> ➤ Edit Curb Return Profiles

Command Line

EditIntersectionCurbReturnProfiles

Dialog Box

Intersection Curb Return Profile Parameters Dialog Box (page 1934)

# Editing Intersection Dynamic Profile Rules

# Widening Turning Lanes for Incoming and Outgoing Roads

# Changing Corridor Assembly Frequency

You can change the frequency of assemblies applied in corridors within intersection objects.

The frequency that assemblies are applied to corridors in an intersection is specified at intersection creation, and is controlled at the command settings level.

You can specify different frequencies for tangent, circular arcs, or spirals segments separately. Once the intersection is created, you can edit the corridor properties to change frequencies as needed. For example, suppose originally the curb return fillet is from station 1+00 to 3+00.

Then suppose that you change the frequency of assembly insertion for one or more of the corridor regions; for example, quadrant 2. Also perhaps you may add manual stations, for example, at station 2+05 and at 2+15 in the curb return fillet section of the corridor in quadrant 2.

Now suppose that you edit the intersection geometry so that the curb return fillet radius at quadrant 2 is increased. Suppose that due to this change, the curb return fillet station changed from 0+50 to 3+50 because of the increased length of arc. In this case, the intersection object will do the following:

■ Corridor region station for that curb return fillet area (in quadrant 2) will be updated as 0+50 to 3+50.

■ Frequencies for this region will be based on the newly set values.

■ Manual stations at 2+05 and 2+15 will still be in the list.

■ All other regions in other quadrants and region types will still have the frequencies set during the intersection object creation.

If the corridor is set to rebuild automatically, then the corridor will be updated with the changes noted above, including manual stations (which would remain at their previous station values).

If one of the manual stations is no longer part of the new curb return fillet area, then that manual station is removed from the list.

If the corridor is set with Rebuild Automatic turned off, then the intersection will still change the corridor regions to the new stations. However, the corridor is not rebuilt unless "Rebuild" is selected on the corridor object, either in the drawing, or in Prospector.

**NOTE** The above steps describe behavior when the corridor is set to rebuild automatically. If it is not set to rebuild automatically, this behavior occurs only when you explicitly click the Rebuild button.

**To change assembly frequency**

1 In Toolspace, on the Settings tab, expand the Intersection collection. Expand the Commands collection.

2 Right-click the CreateIntersection command and click Edit Command Settings.
   The Edit Feature Settings - Intersection dialog box (page 1919) is displayed.

3 Expand Assembly Insertion Defaults.

4 Click in the Value cell for the following frequency options, and specify the desired values: Frequency Along Tangents, Frequency Along Curves, Frequency Along Spirals, Frequency Along Profile Curves.

5 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Intersection ➤ Commands ➤ Right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - Intersection (page 1919)

# Updating Corridor Regions in Intersections

You can update the corridor regions in an intersection manually or automatically.

If the intersection definition is out-of-date, 🛡 is displayed next to the intersection in the Prospector tree. This indicates that the corridor regions in the intersection need to be updated.

When you update corridor regions in an intersection, any manual edits that were made to the regions or targets within the intersection corridor are lost.

You can update corridor regions in an intersection manually, or you can specify that the intersection object update corridor regions automatically. When you configure the intersection object to update corridor regions automatically, a check mark is displayed next to the Update Regions Automatically option on the Toolspace shortcut menu.

```
  Update Regions and Rebuild Corridor
✔ Update Regions Automatically
```
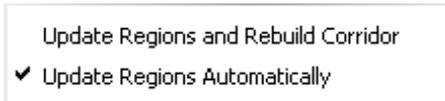
The following two options are available for updating corridor regions in an intersection:

| Options | Description |
| --- | --- |
| Update Regions and Rebuild Corridor | Updates the corridor regions in an intersection object and rebuilds the corridors that are included in the intersection object. |
| Update Regions Automatically | Specifies that the intersection object automatically updates the corridor regions and rebuilds the corridors in the intersection object whenever certain types of changes are made to the intersection. |

When the Update Regions and Rebuild Corridor command is invoked, the corridor region stations in the intersection are updated (recalculated) in order to stay in synchronization with the intersection area geometry. The corridor properties are updated to reflect these new region station values.

**To manually update the corridor regions in an intersection**

■ In Toolspace, on the Prospector tab, expand the Intersections collection. Right-click the intersection. Click Update Regions and Rebuild Corridor.

**NOTE** If the intersection definition is out-of-date, 🛡 is displayed next to its node in the Prospector tree.

**To automatically update the corridor regions in an intersection**

■ In Toolspace, on the Prospector tab, expand the Intersections collection. Right-click the intersection. Click Update Regions Automatically.

**NOTE** When you toggle this option on, if AutoCAD Civil detects that changes have been made to the intersection that require the corridor regions to be updated, then the update occurs immediately when you toggle the option on.

A check mark is displayed next to the Update Regions Automatically option on the Toolspace shortcut menu, indicating that this option is turned on.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Intersections ➤ right-click <intersection name> ➤ Update Regions and Rebuild Corridor or Update Regions Automatically

Object Shortcut Menu

Update Regions and Rebuild Corridor or Update Regions Automatically

# Recreating Corridor Regions in Intersections

You can recreate corridor regions in an intersection after intersection geometry has changed, or after you have applied different assemblies.

You may need to recreate corridor regions if intersection components have been edited. For example, if you change the radius or length of curb return alignments, you must recreate corridor regions in the intersection area to ensure that no geometry gaps or overlaps exist.

Another scenario for recreating corridor regions is as follows:

You may want to discard all changes made to the intersection corridors, and recreate the corridors again using the original intersection creation logic.

For example, if you created an intersection object using a particular assembly set (page 2467), and then made changes to that intersection, such as changing curb return corridor stations, frequencies, and assemblies. You may also have changed the extents of the entire length of the primary road on either side of the intersection. Perhaps next you change the horizontal geometry and curb return parameters. If the corridor is set to rebuild automatically, it will update automatically. Otherwise, at this point the corridor will be displayed as out-of-date in Prospector.

Now you may want to discard all those changes and either reapply the original assemblies specified during intersection creation, or apply different assemblies. If so, you can use the Recreate Corridor Regions command to do this.

When you select the Recreate Corridor Regions command, the Intersection Corridor Regions dialog box is displayed. On this dialog box, you can change parameters if desired, including assemblies to apply, or you can leave the parameters on this dialog as specified, and click Recreate to recreate the corridor regions in the intersection object according to the parameters, and assemblies, that were specified when this intersection object was originally created.

The Recreate Corridor Regions command performs the following actions:

- All corridor regions in the selected intersection are deleted, and are recreated according to the parameters that were specified on the Intersection Corridor Regions dialog box.

- Any user-defined corridor regions within the extents of the intersection are deleted.

- A new set of corridor regions is created according to the assembly insertion frequency that is specified in the command settings.

Because this command deletes and recreates the corridors in the intersection, if you made any changes to the corridors in the intersection object, those changes will be overwritten (discarded) when the new corridors are created.

**To recreate corridor regions in an intersection**

1   In the drawing, select the intersection object.

2   Do one of the following:

   - In the Intersection tab on the ribbon, click the Modify panel ➤ Recreate Corridor Regions ⬚ .

   - Right-click and select Recreate Corridor Regions.

3   On the Intersection Corridor Regions dialog box, click Recreate.

**Quick Reference**

Ribbon

Click Intersection tab ➤ Modify panel ➤ Recreate Corridor Regions 

Toolspace Shortcut Menu

Prospector tab: Intersection ➤ right-click <intersection name> ➤ Recreate Corridor Regions

Object Shortcut Menu

Recreate Corridor Regions

# Deleting Intersection Objects

You can delete an intersection object, erasing it from the drawing and removing it from the Intersections collection in the Prospector tree.

When you delete an intersection object in the drawing, the following are deleted:

■ the intersection object, and its label, in the drawing

■ the intersection object in the Intersections collection in Prospector

The following components associated with an intersection object are *not deleted* when you delete an intersection object:

■ the following 2D geometry components associated with the intersection object: intersecting alignments, offset alignments, curb return alignments

■ the following other components associated with the intersection object: offset profiles, curb return profiles

■ If you created a new corridor object inside the intersection area, this corridor object is *not deleted* when you delete an intersection object. You must delete this corridor object manually, if desired.

You can use the following methods to delete an intersection object:

■ In the drawing, select the intersection object, and delete or erase it.

■ If you delete one or both of the intersecting alignments that create the intersection, then the intersection object is automatically deleted.

■ In the drawing, if you move one or both of the intersecting alignments that create the intersection so that the alignments no longer intersect, then the intersection object is automatically deleted.

**Deleting Intersection Object Dependent Components**

It is important to understand the following behavior associated with deleting objects that intersection objects depend on.

■ **Centerline Profiles**: If any of the centerline profiles associated with the intersection are deleted, then the curb return profiles will become static (no longer a dynamic profile). Accordingly, the intersection corridors will be updated. This behavior is similar to how a corridor reacts to its profile grade line, or when a centerline profile is deleted. In this scenario, the intersection object still exists (is not deleted), with just horizontal geometry.

■ **Offset Alignments**: If an offset alignment associated with an intersection object is deleted, and that offset alignment has a curb return alignment attached to it, the curb return attached to that offset alignment

is not deleted; however, the curb return alignment is no longer dynamic to the intersection object. It becomes a static curb return alignment, meaning that if you move the intersection object, the now static curb return alignment does not move. The same is true for the profile associated with the curb return alignment. It becomes a static (non-dynamic) profile.

- **Curb Return Alignments**: If the curb return alignments or profiles are deleted (using AutoCAD Delete), then the intersection area corridors based on those baselines are also deleted. If only the profile is deleted, then the corridors based on those profiles are moved to zero elevation. The intersection object still exists. Deleting the curb return does not automatically delete the profile associated with it; however, if you delete the curb return, the curb return profile becomes static.

- **Assemblies**: If any of the assemblies used the intersection are deleted, and the corridor is set to "Rebuild Automatic", then the corridor regions that use the deleted assemblies are set to <None>. This creates an empty corridor even though corridor regions still exist. You can use the Recreate Corridor command to download the assemblies from the source location into the current drawing, and this recreates the corridor regions.

**To delete an intersection object**

- In the drawing, select the intersection object, and use the AutoCAD Delete or Erase command.
  The intersection object, and its label, are erased from the drawing. The intersection object is also removed from the Intersections collection in the Prospector tree.

# Labeling Intersections

You can add labels to intersection objects.

There are three ways to add labels to intersection objects:

- during intersection object creation

- using the AddLabel command

- using the AddIntersectionLabel command

By default, the standard intersection object label contains the name of the intersection object, the names of the intersecting alignments, and the station value locations where the alignments intersect.

There is no placement option available for an intersection label. The label is always placed at the intersection point. If that point is already labeled, then a second label is placed on top of the existing label.

For more information, see Understanding Labels in AutoCAD Civil (page 1466).

**To label intersection objects**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Add Intersection Label 🏷️ .

2 Select the intersection object to be labeled.

## Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Add Intersection Label 🏷️

Command Line

AddIntersectionLabel

# Assembly Sets

An assembly set is a group of assemblies used to create a corridor in an intersection object.

Assembly sets are used only with intersection object creation. A default assembly set is included with AutoCAD Civil. You can create an intersection using this default assembly set, or you can create your own custom assembly sets to suit the needs of specific intersection design.
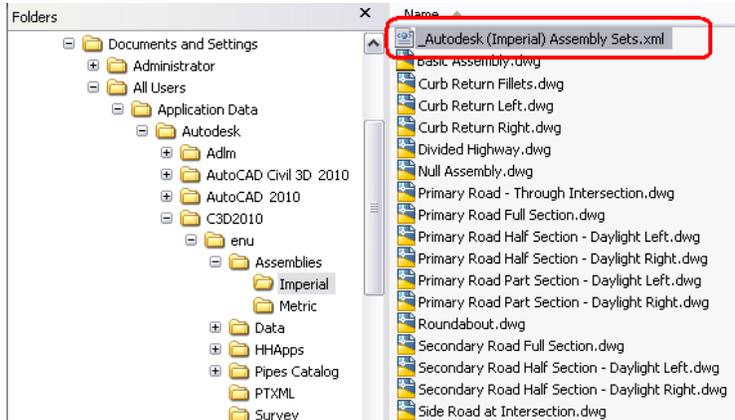


When you apply the assembly set, and create or update the intersection object corridors, the referenced assembly objects are imported into the current drawing, if they are not already there.

The following table describes the components that are associated with an assembly set:

| Component | Description |
| --- | --- |
| assembly set | A group of assemblies that have been designed for use in specific type of intersection design. An assembly set is defined in an assembly set file. |
| assembly set file | An .xml file containing information that defines the assemblies that are included in an assembly set. An assembly set file contains the names of the assembly objects, the location (path) where the individual assembly drawings exist, and other information about the assemblies included in the set. This .xml file serves primarily as a pointer (reference mechanism). It does not contain the actual assembly drawing data. |
| assembly object names | The name of each assembly object referenced in the set is maintained in the assembly set file. Note that the assembly object data itself is not contained in the assembly set file, just the information that enables AutoCAD Civil to locate the assembly object data (a reference to the assembly name, and the name of the drawing file that contains the assembly object.) |
| assembly drawings | Each assembly object referenced in an assembly set is also saved to an assembly drawing file. An assembly drawing file contains a single assembly object. The assembly drawing file typically has the same name as the assembly object. For example, the assembly drawing file for an assembly object named Secondary_Road_Half_Section _Daylight_Left is named Second- |

| Component | Description |
|---|---|
| | ary_Road_Half_Section _Daylight_Left.dwg. The assembly drawing file associated with each assembly object referenced in the assembly set is also stored (referenced) in the assembly set. |

The fist time you use the Create Intersection wizard, it is recommended that you accept and use the assembly set that is referenced by default on the wizard. As you become more familiar with creating intersections, you will want to create your own customized assembly sets. For instructions see, To Create an Assembly Set (page 1458).

The assembly set that is included with AutoCAD Civil is typically located by default in the following location:

*C:\Documents and Settings\All Users\Application Data\ Autodesk\Civil2010\enu\Assemblies\Imperial (or Metric)*

**NOTE** If the assembly set file references an assembly name that cannot be found, then a null assembly (page 2478) will be used to in its place to create the intersection object. For more information, see Null Assemblies (page 1460).

The following illustration shows the contents of the default assembly set file that is included with AutoCAD Civil.
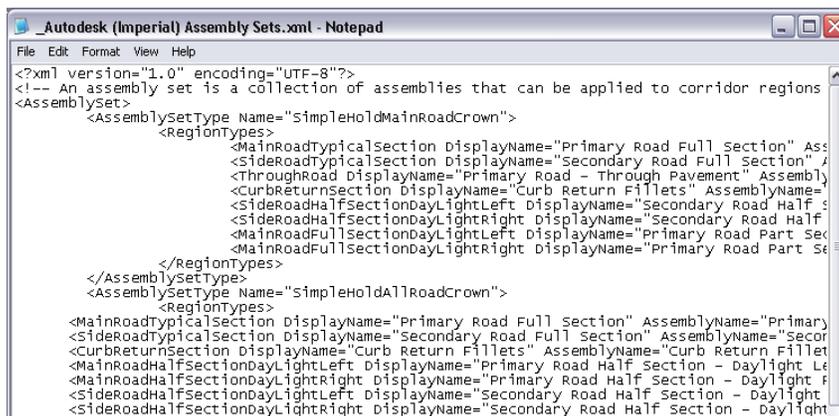


### Modifying Assemblies Referenced in Assembly Sets

It is important to note that if you modify an assembly that is referenced in an assembly set, and that is already used in the current drawing, and you do not change the name of the modified assembly, this does not change the intersection area corridor definition.

In such cases, if you try to create a new intersection, or recreate an existing intersection, the intersection corridor will continue to use the assembly of the same name in the current drawing, and will not use the modified assembly that is referenced in the assembly set folder.

If you want to apply the updated assembly that is referenced in the assembly set to the intersection corridor, you must delete (or rename) the assembly with the same in name in the current drawing first.

**To create an assembly set**

1   Ensure that the assemblies that you intend to save to the new assembly set exist in the current drawing.

 You can perform this procedure using the Create Intersection wizard, or by editing an existing intersection object.

2   Do one of the following:

 ■ **Using the Create Intersection Wizard**: You can create an assembly set during intersection creation, using the Corridor Regions page of the Create Intersection wizard. To do so, start the CreateIntersection comand, select the new assemblies in the Assemblies To Apply section of the

Corridor Regions page, and then proceed to Step 3 below. If desired, you can click Cancel on the Create Intersection wizard after you have saved the new assembly set. This allows you to create a new assembly set without creating a new intersection object.

- **Edit an Existing Intersection**: Select an existing intersection, and then click Recreate Corridor Regions on the Modify panel of the Intersection contextual tab. On the Intersection Corridor Regions dialog box, select the assemblies in the Assemblies To Apply section of this dialog box.

3 After selecting the desired assemblies, click Save As a Set.

4 On the Select Assembly Set File dialog box, enter a name for the new assembly set file, and then click Save.

---

**NOTE** At this point, the new assembly set file is created. You may or may not want to proceed with applying the new assembly set to the current intersection. If you do want to apply the new assembly set to the current intersection, follow the next step.

---

5 To apply the assembly set to the intersection, do one of the following:
- If you selected the assemblies from the Corridor Regions page of the Create Intersection wizard, click the Browse button, select the newly created assembly set file, and then click Create Intersection.

- If you selected the assemblies from the Intersection Corridor Regions dialog box, click the Browse button, select the newly created assembly set file, and then click Recreate.

**To apply a different assembly set to an intersection**

1 In the current drawing, create the desired assemblies. You can edit the existing assemblies, or create new ones from scratch.

2 Do one of the following:
- If you are creating an intersection, on the Corridor Regions page of the Create Intersection wizard, select the new assemblies in the Assemblies To Apply section of this dialog box.

- On an existing intersection, select the intersection, and then click Recreate Corridor Regions on the Modify panel of the Intersection contextual tab. On the Intersection Corridor Regions dialog box, select the assemblies in the Assemblies To Apply section of this dialog box.

3 After specifying the desired assemblies, click Save As a Set.

4 On the Select Assembly Set File dialog box, specify the name and location for the assembly set file, and click Save. If desired, you can select an existing assembly set file and save (update) it with the information about the newly specified assemblies.

5 Click Browse to navigate to and select the desired assembly set file.

6 Use the drop-down arrow on the Select Assembly Set To Import field to ensure that the desired assembly set file is specified on this dialog box.

7 To apply the new assembly set to the intersection, do one of the following:
- If you are creating a new intersection, click Create Intersection on the Corridor Regions page of the Create Intersection wizard.

- If you edited an existing intersection, click Recreate on the Intersection Corridor Regions dialog box.

# Null Assemblies

A null assembly is created in an intersection object during creation when an assembly name referenced in the assembly set file cannot be found.

AutoCAD Civil automatically creates a null assembly when you create an intersection that references an assembly name that cannot be found in any of the assembly drawings referenced in the selected assembly set (page 2467) file. The null assembly serves as a placeholder object so that the intersection object can still be created even though an assembly cannot be found.

If the assembly name is not found in the assembly drawings referenced by the assembly set file, AutoCAD Civil then searches for the assembly name in the current drawing. If the assembly is found in the current drawing, then AutoCAD Civil proceeds with creating the intersection using that assembly, and creating the external assembly drawing for the assembly, if one does not already exist.

A null assembly allows AutoCAD Civil to continue with intersection creation even though all of the referenced assemblies are not found.
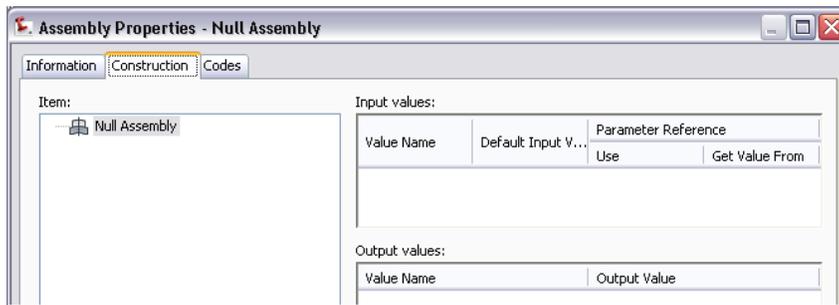




After the intersection is created, you can assign appropriate assemblies, or a different assembly set. If you do, you will need to recreate the intersection corridor regions, as well as rebuild the intersection, to apply the new assembly or assemblies to the intersection. You may also need to rebuild corridors. For more information on recreating intersection corridor regions, or on rebuilding an intersection, see Updating Corridor Regions in Intersections (page 1452) and Recreating Corridor Regions in Intersections (page 1454).

# Intersection Command Reference

You can use commands to quickly access intersection functionality.

The following table lists the intersection-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AddIntersectionLabel | Adds a label to an intersection object (page 1456) |
| CreateIntersection | Creates an intersection (page 1430) |
| EditIntersectionProperties | Displays the Intersection Properties dialog box which lets you edit Intersection object properties (page 1441) |

| Command | Description |
|---|---|
| EditIxCorridorRegions | Edits corridor regions in an intersection (page 1452) |
| EditIxCurbReturns | Edits curb returns in an intersection (page 1447) |
| EditIxCurbReturnProfiles | Edits curb return profiles associated with an intersection object (page 1450) |
| EditIxOffsetProfiles | Edits offset alignment profiles associated with an intersection object (page 1449) |
| EditIxOffsets | Edits offset alignments in an intersection (page 1445) |
| EditIxSideRoadProfile | Edits side road profiles associated with an intersection (page 1448) |

# Labels and Tags

# 33

You can annotate AutoCAD Civil objects with labels that update dynamically when the drawing objects change.

Labels can contain single or multiple lines of text, blocks, ticks, lines, and direction arrows.

Labels have two modes: label mode and tag mode. Use tags to mark drawing objects with numbers so you can insert detailed information in tables.

In AutoCAD Civil, tag settings are a drawing-wide setting. See Creating Label Tags (page 1544) for more information.

Labels and Tables Workflow

Label Styles Best Practices

## Label Objects

AutoCAD Civil includes independent label objects. Most labels in AutoCAD Civil are implemented as label objects that reside on their own layer.

**NOTE** Points, surface watershed, and all corridor labels are not object type labels. They are sub-entities of a parent object and their properties are managed in the Label Properties dialog box.

- Label objects are not highlighted when the parent object is selected as they are separate independent objects.

- Labels can include references to other AutoCAD Civil objects by including Referenced Text (page 1509) components.

- You can label external reference (xref) data objects. When you insert xref objects from a source drawing into your current drawing, you can annotate them as you would other objects. For more information, see Labeling External References (Xrefs) (page 1531).

- You can use alignment label styles to label superelevation critical points (page 877).

- Label text can be overridden by using the Edit Label Text (page 1535) command on the shortcut menu.

- Group Label Properties (page 1533) such as such as layer, linetype, and color are changed with the AutoCAD Properties palette. You have the ability to simultaneously edit group labels with the Properties palette.

- Label object properties are separate from the parent object. Use the Label List command (page 1486) to display the label properties (including its designation as a single or group label object).

- Label visibility (page 1487) can be controlled with label style settings or through layer management.

- New label styles can reference the layer in the style, or the layer that the label(s) resides on. This gives you the option of managing labels with styles (page 1492) or the use of layers.

- Labels that are part of a group can be selected individually and changed using "Ctrl+click" selection (page 1534).

- A label's default layer (page 1488) can be specified and changed with the Properties palette, or other AutoCAD layer manipulation tools.

- Alignment label styles are edited in the Alignment Labels Dialog Box (page 1739).

- Use the Mapcheck Analysis command to determine values from label objects based on the precision of the annotation of the label object. Click Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck 🗹 . For more information, see Performing a Mapcheck Analysis (page 1608).

- This release of AutoCAD Civil features label leader line enhancements for improved control and flexibility when grip editing labels. See Editing Line, Arc, and Polyline Labels with Grips (page 1538), and Managing Leader Vertices (page 1520) for more information.

- Use the Label Stagger options to avoid densely populated graph views. These options allow you to stagger labels for profile views, section views, and data bands for improved visibility. For more information, see Staggering Profile and Section Labels (page 1527) and Staggering Data Band Labels (page 1528).

- Data band labels are now object labels that can be edited, dragged, and reset like other label objects. See Editing Data Band Labels (page 1529), Deleting Band Labels (page 1545), and Restoring Band Labels (page 1546) for more information.

- This release of AutoCAD Civil include projected object label styles for profile and section views. See Adding Profile View Labels (page 1037)and Using Section View Labels (page 1139)for more information.

The label objects are listed in the table below. Note that the label objects include two distinct object types: group and single labels.

Group label objects and single label objects behave differently with respect to screen interaction and AutoCAD object management. The selection of the group label objects is limited to the group (the entire group is highlighted when selected), which means the context menu no longer applies to the perceived selected label.

This group selection functionality allows you to simultaneously edit an entire group of labels. As group type labels have an object controlling the creation and application of the label within a group, the AutoCAD Properties palette lists information regarding the annotated object.

Single type labels do not list this information, but both types display Standard AutoCAD properties and Label properties with the Properties palette. See Modifying Labels in a Drawing (page 1533) for more information.

The label objects are listed in the table below and are identified as either group or single labels.

| Feature / Label Type | Group or Single Label |
| --- | --- |
| Note | Single |
| Segment | Single |
| Surface Slope | Single |
| Surface Spot Elev | Single |
| Surface Contour - Major, Minor, User | Group (no sub-entity selection) |

| Feature / Label Type | Group or Single Label |
|---|---|
| Alignment Major Stations | Group |
| Alignment Minor Stations | Group |
| Alignment Geometry Point | Group |
| Alignment Station Equation | Group |
| Alignment Design Speed | Group |
| Alignment - Profile Geometry Point | Group |
| Alignment Station Offset - Fixed Point | Single |
| Alignment Station Offset | Single |
| Alignment Line | Single |
| Alignment Curve | Single |
| Alignment Spiral | Single |
| Alignment Tangent Intersect | Single |
| Parcel Line | Single |
| Parcel Curve | Single |
| Parcel Area | The first area label is part of the parcel object (to select the parcel). Additional area labels can be added, but they are single. |
| Profile Major Stations | Group |
| Profile Minor Station | Group |
| Profile Horizontal Geometry | Group |
| Profile Grade Breaks | Group |
| Profile Line | Group |
| Profile Sag Curves | Group |
| Profile Crest Curves | Group |
| Profile View Depth | Single |
| Profile View - Station Elevation | Single |
| Sample Line | Group |
| Section Major Offset | Group |

| Feature / Label Type | Group or Single Label |
| --- | --- |
| Section Minor Offset | Group |
| Section Grade Break | Group |
| Section Segment | Group |
| Section View Grade | Single |
| Network Pipe in plan | Single |
| Network Pipe in profile | Single |
| Network Pipe in section | Single |
| Network Structure in plan | Single |
| Network Structure in profile | Single |
| Network Structure in section | Single |
| Network Pipe - Spanning pipe | Single |
| Plans Production - View Frame | Single |
| Plans Production - Matchline | Group |
| Profile Data Band | Group |
| Section Data Band | Group |

# Understanding Labels

In AutoCAD Civil, labels and tags are controlled by label styles.

In the Toolspace Settings tree, most feature collections, such as parcels, alignments, and surfaces, contain their own unique label styles. When you define a label style, you determine the purpose of the label and set up a relationship with the object that uses it. Labels always remain associated with the object they are annotating.

General line and curve label styles can be used for parcel segments labels, and alignment line/curve labels. Previously, general line label styles could modify only lines, curves, and polylines. For more information, see Editing Label Styles (page 1482).

The general line label styles are available for the parcel, alignment, grading, and survey labeling commands. Parcel tables now support general line/curve label types. Parcel tables display either general/line curve labels or parcel segment labels. For more information, see Adding Parcel Tables (page 845).

There are three levels of label settings in addition to settings that you define in the label styles themselves. For more information, see The Hierarchy of Label Settings (page 1468).

As you create AutoCAD Civil objects in a drawing, they are labeled automatically using specified label styles. However, after an object is created, you can add more labels manually.

Labels are defined by the following properties:

■ **Location**. Label location in a drawing depends on the object using the label.

For example, a parcel area label is usually placed at the center of the parcel, and surface contour labels are usually placed on the contour line.

- **Appearance and Visibility**. Whether a label is visible, and how it appears in a drawing, can be specified in the label style or by managing the AutoCAD properties. You can define different display settings, such as color, lineweight, and linetype, for each label component.

- **Mode**. Many label styles have two modes: label and tag. Tags are used to identify labeled objects within a table. For more information, see Setting Up Label Styles To Be Used as Tags (page 1485).

- **Behavior**. Labels behave according to their association with objects. For example, if you move an object in a drawing, the label moves with it.
  Also, behavior can also depend on how a label is positioned in a drawing.

- **Placement**. A label is placed in relation to an object or another label style component. For more information, see Anchor Points and Attachment Points (page 1492).

- **Plan-Readable text**. Labels can be placed at any angle, but you can select a plan-readability setting to maintain legibility. For more information, see Using Plan-Readability (page 1489) and Changing Label Insertion (page 1490).

- **Orientation**. Label orientation refers to the rotation angle of the labels in 3D space. For more information about label orientation, see Changing Label Orientation (page 1488).

The commands used to add single and multiple label types from the parcels, alignments and line/curve menus support each of the three object types. For example, if you click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line And Curve ➤ Add Single Segment Line/Curve Label 🏷 and then select a parcel in the drawing, a parcel segment (line or curve) label is created.

The following object type/segments are supported by these commands:

- line

- arc

- polyline line segment

- feature line, line segment

- feature line, arc segment

- parcel line segment

- parcel curve segment

- alignment line segment

- alignment curve segment

- alignment spiral segment

The following table lists what determines the default style/style type when you use the add label command from particular menus.

| | General Line/Curve Menu | Parcel Line/Curve Menu | Alignment Line/Curve Menu |
| --- | --- | --- | --- |
| Create label on line, curve, pline, or feature line | Styles come from line/curve settings (General line/curve style type). | Styles come from Parcel settings (General line/curve style type, or parcel types supported). | Styles come from line/curve settings (General line/curve style type). |

|  | **General Line/Curve Menu** | **Parcel Line/Curve Menu** | **Alignment Line/Curve Menu** |
|---|---|---|---|
| Create label on Parcel Segment | Styles come from line/curve settings (General line/curve style type). | Styles come from Parcel settings (General line/curve style type, or parcel types supported). | Styles come from Parcel settings (General line/curve style type, or parcel types supported). |
| Create label on Alignment Segment | Styles come from line/curve settings (General line/curve style type). When labeling using "Multiple" or "Single," and any spirals are labeled they use the alignment settings for the spiral style. | Styles come from Alignment settings (General line/curve style type, or Alignment types supported). When labeling using "Multiple" or "Single," and any spirals are labeled, they use the alignment settings for the spiral style. | Styles come from Alignment settings (General line/curve style type, or Alignment types supported). |
| Create label on Line, Curve, Pline or Feature Line | Styles come from Add Labels dialog | Styles come from Add Labels dialog | Styles come from Parcel settings (General line/curve style type, or parcel types supported). |
| Create label on Alignment Segment | Styles come from Add Labels dialog. When labeling using "Multiple" or "Single," and any spirals are labeled, they use the alignment settings for the spiral label style. | If current style referenced in label tool is line/curve type, then that style is applied. If style is parcel type, get styles from alignment settings. When labeling spirals using "Multiple" or "Single," and any spirals are labeled, they use the alignment settings for the spiral style. | Styles come from Add Labels dialog. |

# The Hierarchy of Label Settings

You can manage label style settings in a drawing at different levels using the Settings tree in Toolspace.

The hierarchy that label style settings use can be best viewed in the tree structure in Toolspace, shown in the following illustration.

**Edit label settings at the Drawing level, object level (such as Surface),**

**and label-style-type level (such as Contour)**
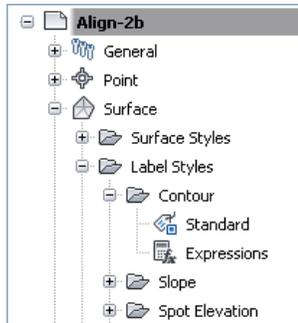


**At each level, right-click and click Edit Label Style Defaults**

Each level of settings controls the characteristics in subordinate levels, unless a setting has been specifically changed or overridden in the subordinate.

At the drawing level you can determine how table tags are numbered. Click the drawing name, right-click and click Table Tag Numbering to access the Table Tag Numbering Dialog Box (page 1971). For more information, see Creating Label Tags (page 1544).
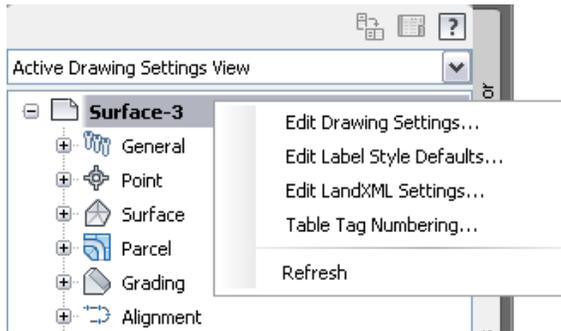


**Tag numbering settings are drawing-wide settings**

# Overview of Label Styles

Use a label style to define the behavior, appearance, and content of labels.

Every label in the drawing has a style associated with it, and edits to the label styles are immediately reflected in the labels in the drawing.

In the Toolspace Settings tree, Label Styles collections contain one or more *types* of label styles that represent a unique aspect of the object using it. The following illustration shows the hierarchy of the label style collection, the label style types, and the individual label styles:

**Parcel Area label style type collection**
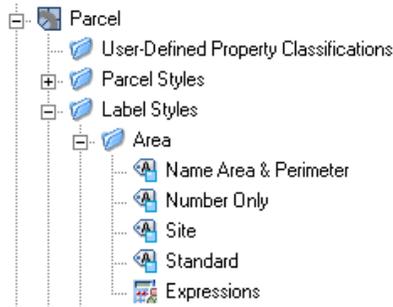


Each label type has a Standard label style. The AutoCAD Civil drawing templates also contain several other label styles you can use.

You can create *child* styles that are based on an existing style. For more information, see Defining Children of Existing Label Styles (page 1478).

## Viewing and Selecting Available Label Styles

Use the Toolspace Settings tree to view available label styles in a drawing.

**NOTE** You can identify a label style in the Settings tree by  next to it. If a label style is in use in the drawing, then  is displayed when the Drawing Item State icons are visible in Toolspace.

**To view available label styles**

1   In Toolspace, click the Settings tab.

> **TIP** From the View Control list, select Active Drawing Labels Only View. Or, if you have more than one drawing open, select Labels Only View.

2   Expand the Settings tree by clicking ⊞ next to a drawing name.

3   Expand a feature collection, such as the Surface collection.

4   Expand the Label Styles collection to display the label style types inherent to that object.
    In the Alignment, Profile, and Section collections, you can also view any label sets (page 1547) for that feature.

5   Expand the label style type collection to view the available label styles.

6   Right-click a label style to access commands for working with the style. You can edit, delete, or copy the style, or create a new child style.

> **NOTE** Right-click the label style type collection and click New to create a new label that is not a child style of an existing style.

OR

1   In the drawing, select the label to edit. Right-click and select Edit Label Style.

2   In the Label Style Control (page 1965) box, click the Select a Style arrow to display a list of available styles.

OR

**1** Select a label, right-click and select Properties or Label Properties.

**2** In the Properties palette, click Line or Curve label style to display a list of available styles.

Select the Edit/Create option to access the Label Style Control (page 1965) dialog box with which you can create a new style, copy the currently selected style, create a child style, or edit the currently selected style with the Label Style Composer.

OR

**1** In the drawing, select the label to edit.

**2** Select the Labels tab ➤ Label Properties drop-down menu ➤ Edit Label Style to access the Label Style Control (page 1965) dialog box.

### Quick Reference

Ribbon

Click Labels tab ➤ Modify panel ➤ Label Properties drop-down ➤ Edit Label Style

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ <label style name>

## Predefined Label Styles in Templates

Use drawing templates provided with AutoCAD Civil to access predefined label styles for each feature.

Use these styles as-is or use them as a basis for creating your own styles.

## Standard Label Styles

By default, all label style types have a Standard label style.

When you create a new drawing without a reference to a template (*.dwt*), the label style types in the Label Styles collections in the Settings tree contain a Standard label style based on default settings. You can use the Standard style as-is, or you can use it as a basis for a new style by changing its properties and renaming it.

You can delete a Standard label style only when it is not referenced in a drawing or if it is not a "parent" label style with a subordinate "child" label style. For more information, see Defining Children of Existing Label Styles (page 1478).

## Working with Label Styles in Toolspace

Use the Settings tree to create or edit label styles.

Right-click either a label style type or a specific label style to display a shortcut menu with options for working with label styles.

| Select this command… | If you want to… |
| --- | --- |
| Right-click label style ➤ New (page 1478) | Create a label style that is based on the settings of a parent label style. This command displays |

| Select this command... | If you want to... |
| --- | --- |
| | the Label Style Composer and contains "Child of..." as the label style name. |
| Right-click label type ➤ New (page 1477) | Create a label style that is not a child of an existing label style. The new label style is a parent-level style and obtains its defaults from the label type settings. |
| Right-click label style ➤ Edit (page 1473) | Edit a label style in the Label Style Composer. |

## Using Label Styles From Other Drawings

Use a label style from any open drawing by dragging the style from one drawing to another in the Settings tree. You can also drag the style from the Settings tree into the drawing window to copy it into that drawing.

When you attempt to copy a named label style to a location where a matching named label style exists, you are prompted with a warning, and you can choose one of the following options:

■ **No**: Leaves the existing style and cancels the operation.

■ **Yes**: Copies the style to the new drawing and renames the style using the format <source drawing name>.<style name>.1

### Using Child Label Styles From Other Drawings

If you copy a child label style to another drawing, its parent style is also copied.

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**TIP** Because you can only drag one label style at a time into another drawing, it is more efficient to set up your commonly used label styles in a DWT file that you can use for each new drawing you create.

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### To use label styles from other drawings

1  Open the drawing that contains the styles you want to copy (Drawing1, for example), and the drawing you want to copy styles to (Drawing2, for example).

2  In Toolspace, click the Settings tab.

3  From the View Control list, select Labels Only View to see the labels in all open drawings.

4  In the Settings tree, expand the Drawing1 collection and select the label style that you want to copy.

5  Drag and drop the label style to Drawing2 in the Settings tree. You must drag the style over the drawing name in the Settings tree before releasing the mouse button.

    ■ A valid drag and drop operation will display a ⬚ cursor.

    ■ An invalid drag and drop operation will display a ⊘ cursor.

## Previewing Label Styles

Use the preview window to view how a label style will appear in the drawing. When you make modifications to the label style, the preview window updates.

The preview window is located on the General, Layout, and Dragged State tabs of the Label Style Composer (page 1473). You can select a different preview by selecting it in the Preview list.

To change the view in the Preview window, you can use AutoCAD commands on the shortcut menu, such as Pan and Zoom.

Previews are actually stored as DWG files in the \Data\Preview folder. If you would like to add your own preview drawings that contain specific label styles, you can place the drawings in the Preview folder.

**NOTE** Do not rename the Preview folder or its sub-folders. Any changes to the folder structure prevents AutoCAD Civil from accessing the preview drawings.

**To add preview drawings to the Preview folder**

1 Create a new drawing, create one or more label styles, and label an object with the label styles you want to preview.

2 Click App Frame ➤ Save As.

3 Navigate to the Data\Preview folder and save the drawing file.

**NOTE** The \Data folder path is set up during installation.

**To view a preview drawing in the Preview pane**

1 On the General, Layout, or Dragged State tabs of the Label Style Composer (page 1942), click the Preview arrow in the upper-right corner of the dialog box to display a list of preview drawings.

2 Select the preview drawing you want to view.

3 Right-click in the preview pane to access commands you can use to change the display of the preview drawing.

# Overview of the Label Style Composer

Use the Label Style Composer to edit a style or to set up a new label style

**Access the Label Style Composer by selecting any of the following commands:**

■ Right-click an existing style and click Edit.

■ Right-click an existing style and click New.

■ Right-click a label style type and click New.

**NOTE** You can also access the Label Style Composer from the Label Style Control, which is available in many dialog boxes. For more information, see Label Style Control (page 1965).

**Access the Label Style Composer from the Ribbon**

1 Select the label. Click Labels tab ➤ Modify panel ➤ Label Properties drop-down ➤ Edit Label Style  .

2 In the Label Style Control, click Edit Current Selection.

The Label Style Composer contains the following five areas for settings:

**Information tab (page 1942)** Defines the name and description of the label style.

**General tab (page 1942)** Defines general display and behavior of the label, such as layer and visibility settings. For more information, see Managing General Properties for Label Styles (page 1486).

**Layout tab (page 1944)** Defines specific components of a label, and their display, behavior, and relationship to the feature that uses them. For more information, see Managing Layout Properties for Label Styles (page 1492).

**Dragged State tab (page 1958)** Specifies how label text displays when it is dragged away from its original position. For more information, see Managing Dragged Properties for Label Styles (page 1517).

**Summary tab (page 1959)** Displays all settings and overrides in a label style, as well as parent/child relationships. For more information about working with overrides, see Managing Overrides in Label Style Dialog Boxes (page 1960).

# Scaling Labels in Drawings

Labels are scaled relative to the drawing scale in model space, and are scaled to the assigned text height in paper space, regardless of the viewport scale.

**Model space scaling**

In model space, label text height is controlled by the drawing scale and the text height as specified in the label style.

For example:

- In a 1:50 scale drawing, with the label text height 0.1, the labels are created at a height of 5'.

- In a 1:100 scale drawing, with the label text height 0.1, the labels are created at a height of 10'.

**Paper space scaling**

In paper space, label text always appears at the same height regardless of the paper space viewport scale.

For example, label text height set to 0.1 appears as 0.1 inch on the plotted sheet, regardless of viewport scale or zoom level.

You must use the REGENALL command to update the label display onscreen when you either change the paper space viewport scale or zoom.

**Scaling of Xrefs**

Labels in drawings that you xref also respect the scale of paper space viewports, so you can create project documentation in paper space at the correct scale for several drawings at the same time.

**NOTE** In order to label an xref object, the scale of the inserted xref is must be 1,1,1.

**Recommendations for managing AutoCAD Civil labels in a viewport**

When working with labels in a viewport, do not use the Annotation Scale at the bottom of the drawing window as this will change the viewport scale.

When working with labels in a viewport, you should use one of the following methods:

- Use the Standard Scale in the View Port Properties Palette window. This will not change the Annotation Scale.

- Use the zoom command in the viewport to get the desired viewport zoom scale.

# Defining Label Settings

You can define default label settings at three different levels. Each level controls default characteristics in subordinate settings and in the label styles.

At subordinate levels you can override the default settings. However, you can lock the default settings so that they cannot be changed.

## Default Settings for All Label Styles in a Drawing

Label style defaults that you set at the drawing level can affect every label style in the drawing.

**To specify default label style settings for a drawing**

1   In Toolspace, on the Settings tab, right-click a drawing name, and click Edit Label Style Defaults.

2   In the Edit Label Style Defaults - Drawing (page 1941) dialog box, define defaults for all labels in a drawing by changing the settings in the Value column.

> **TIP** Using the Visibility setting in the drawing-level label settings is a quick way to turn off all labels in a drawing.

If you want a different setting for an individual label style, you can override the default in a subordinate setting.

3   Optionally, lock any setting by clicking ⬔ to change it to 🔒 .

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

   Settings tab: Right-click <drawing name> ➤ Edit Label Style Defaults

Dialog Box

   Edit Label Style Defaults - Drawing (page 1941)

## Default Settings for All Label Styles in a Feature

Label style defaults that you set at the feature level can affect every label style defined for that specific feature.

**To specify default label settings for all label styles in a feature**

1   In Toolspace, on the Settings tab, right-click a feature collection and click Edit Label Style Defaults.

2   In the Edit Label Style Defaults <Feature Name> (page 1941) dialog box, define defaults for all label styles belonging to the feature by changing the property settings in the Value column.

When you change a value at the current level, the check box ☑ is selected in the Override (page 1960) column.

**3** Optionally, lock any setting by clicking 🔓 to change it to 🔒 .

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ Right-click <object collection> ➤ Edit Label Style Defaults

Dialog Box

Edit Label Style Defaults <Feature Name> (page 1941)

# Default Settings for All Labels in a Label Type

Label style defaults that you set at the label-style-type-level can affect every label style defined for that specific label style type.

**To specify default settings for all labels in a label type**

**1** In Toolspace, on the Settings tab, right-click a label style type and click Edit Label Style Defaults.

> **NOTE** Label style types are the lowest-level collection within the Label Styles collection. For example, under Alignment, expand Station to display the label style types Major Station, Minor Station, Geometry Point, and so on.

**2** In the Edit Label Style Defaults <Style Type> (page 1942) dialog box, define defaults for all label styles belonging to that label type collection by changing the property settings in the Value column.

When you change a value at the current level, the check box ☑ is selected in the Override (page 1960) column.

**3** Optionally, lock any setting by clicking 🔓 to change it to 🔒 .

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ Right-click <label style type name> ➤ Edit Label Style Defaults

Dialog Box

Edit Label Style Defaults <Style Type> (page 1942)

# Changing Command Settings for Labeling Lines, Arcs, and Notes

Specify the default styles to use when inserting Note and Line and Curve labels.

**To specify default styles for Note and Line and Curve Labels**

**1** In Toolspace, on the Settings tab, expand the General collection.

**2** Expand the Commands collection.

**3** Right-click a command item and click Edit Command Settings.

**4** Expand Default Styles.

**5** Specify the styles for each label type.

**6** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ General ➤ Commands ➤ Right-click <command name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings - General Dialog Box (page 1799)

# Creating and Editing Label Styles

You can create new label styles that are derived from existing styles and styles that are independent of existing label styles. When you edit a label style, all labels in the drawing that use that style are updated.

**NOTE** Label style names are case-sensitive. For example, the label name "Standard" is not the same as "standard".

# Creating New Label Styles

Define the settings for a new or existing label style by using the Label Style Composer.

## Defining an Independent Label Style

You can define a label style that has no dependencies on another existing label style.

**To define an independent label style**

**1** In Toolspace, on the Settings tab, right-click a label style type and click New.

**NOTE** Label style types are the lowest-level collection within the Label Styles collection. For example, under Alignment, expand Station to display the label style types Major Station, Minor Station, Geometry Point, and so on.

**2** In the Label Style Composer, click the Information (page 1942) tab and enter a name in the Name box.

**3** Use the other tabs in the Label Style Composer (page 1942) to change the settings.

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ right-click <label style type name> ➤ New

Dialog Box

Label Style Composer (page 1942)

## Defining New Label Styles by Copying Existing Label Styles

Create a copy of a label style.

**To copy label styles**

1   In Toolspace, on the Settings tab, right-click a label style and click Copy.

2   In the Label Style Composer, click the Information (page 1942) tab and enter a name in the Name box.

3   Use the other tabs in the Label Style Composer (page 1942) to change the settings.

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Copy

Dialog Box

Label Style Composer (page 1942)

## Defining Children of Existing Label Styles

You can define a new child label style that derives its default settings from an existing label style, or parent.

**To create a child of a label style**

1   In Toolspace, on the Settings tab, right-click a label style, and click New.
    In the Label Style Composer, "<name of parent style> [Child]" is displayed in the title bar.

2   Click the Information (page 1942) tab, and enter a name in the Name box.

3   Use the other tabs in the Label Style Composer (page 1942) to change the settings.

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ New

Dialog Box

Label Style Composer (page 1942)

## Automatically-Created Child Styles

To be displayed in a table, the labels must be converted to tag display mode. Tag and label mode are style-wide settings. However, when you manually select labels that are in label mode, AutoCAD Civil generates a child style that is set to tag mode and changes only the selected label to this new child style.

**Selecting Labels to Add to a Table**

When you insert a table, you can specify the contents of the table in two ways:

■ The table can contain all labels of a particular label style.

■ You can select the labels to add to the table from the drawing.

When you use the second method, you are prompted to create a child style if the selected labels are currently in Label display mode.

The following illustration shows the Selection section of the Add Table dialog box:
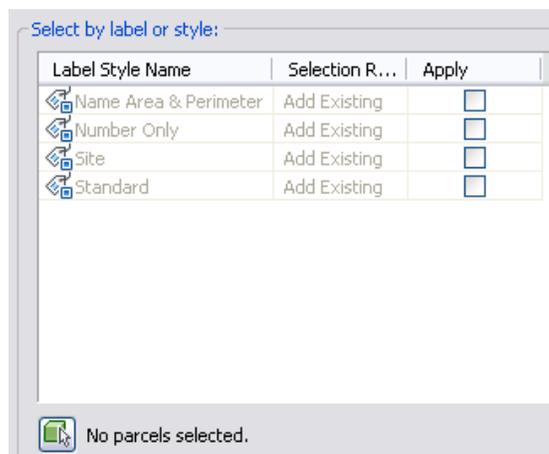


To add all labels of a particular label style name to a table, select the Apply check box next to the label style name. When you use this method, all the labels are switched to tag mode, and a child style is not created.

To add selected labels of a particular style to a table, click and click the labels. When you select a label in the drawing that is not in Tag display mode, a dialog box is displayed informing you that the label is not currently in tag mode and asking whether you want to create a new child style.

Click Yes to continue (if you click No a child style is not created and the label is not added to the table). Clicking Yes creates a child style of the current label style in the format <style name>.Tag.1.

If you want to avoid generating child styles when creating a table, then you can do either of the following:

■ Always use the style-based selection method when creating tables. You can specifically set up "tag" label styles to use for this purpose, and use the tag label styles to label only those entities you want to insert into the table. For more information, see Setting Up Label Styles To Be Used as Tags (page 1485).

■ Before creating a table, switch the label style display mode to Tag. Then when you use the Select In Drawing method when tags are displayed, child styles are not created. For more information on switching to tag mode, see Displaying Labels as Tags (page 1487).

# Creating Label Styles for Lines, Curves, and Polylines

Line, Curve, and Polyline labels use the styles that are defined in the General ➤ Label Styles collection in the Settings tree.

Polylines use Line label styles for straight segments and Curve label styles for curved segments. You can set up the Line and Curve label styles to label the total length and area of polylines by using the "Overall" label properties.

**To create a line or curve label style**

1  In Toolspace, on the Settings tab, expand the General ➤ Label Styles collection.

2  Right-click the Line or Curve collection and click New.

3  Use the tabs on the Label Style Composer  (page 1942)to set up the label style.

> **NOTE** From the Layout tab, you can add or edit label components to include the overall length of lines and polylines and the total area of polylines. Follow the steps in Adding Text Components to Labels (page 1503) and use the General Overall Length and General Overall Area properties.

4  If you have multiple components in the label style, be sure to specify appropriate Anchor Points and Attachment Points (page 1492) so the components do not overlap. You can use the Preview window to check the label layout.

> **NOTE** Specify the default layer for Line and Curve labels on the Layers tab (page 66) of the Drawing Settings dialog box. You can also change the layer of these labels by using the AutoCAD Properties palette (right-click label and select Label Properties or Properties).

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ General ➤ Label Styles ➤ right-click Line or Curve ➤ New

Dialog Box

Label Style Composer (page 1942)

# Creating Note Labels

Note label styles are inserted at selected points in the drawing, and are not physically attached to specific objects. They can use Referenced Text components to contain references to other objects in the drawing.

The insertion point of a Note label can be used to identify location-based Referenced Text data. For example, in the following illustration, the insertion point determines the station and offset values that appear in the label.

Note insertion point determines offset and station values

Offset = - 50 feet
from station 9+50 on Oak Street

9+00        10+00        11+00

Offset = 100 feet
from station 8+50 on Oak Street

Referenced Text Component
refers to alignment offset, station, and name
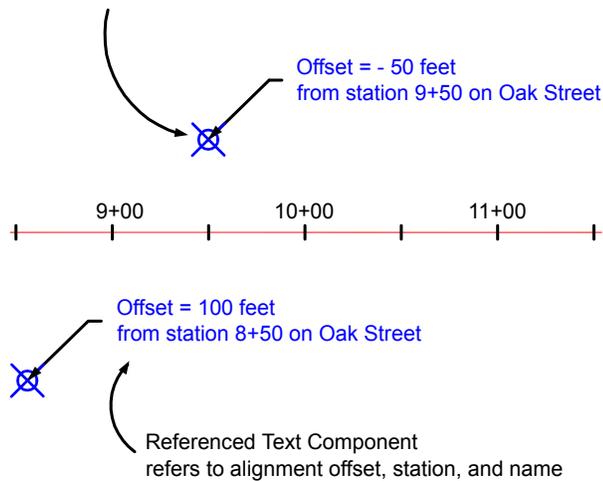
This component would be set up as follows in the Text Component Editor:

```
Offset = <[Offset(Uft|P3|RN|Sn|OF|AP)]> feet
```

```
from station <[Station Value(Uft|FS|P2|RN|AP|Sn|TP|B2|EN|W0|OF)]> on <[Name(CP)]>
```

**To create a note label style**

1   In Toolspace, on the Settings tab, expand the General ➤ Labels collection.

2   Right-click the Note collection and click New.

3   On the Information tab of the Label Style Composer dialog box, specify a name for the new style.

4   Click the Layout tab (page 1944).

5   A default text component is created by default. To change the text string, under Text, click the Contents row. Then click ⋯ .

6   In the Text Component Editor (page 1961), edit the text for the note as needed. You can type directly into the window.

7   Close the Text Component Editor window.

8   If you want to reference objects in the drawing, create a Referenced Text (page 1509) component for each object you want to reference.

For example, you can create a note label with two Referenced Text components, each which refers to an alignment name. You can then insert that label at the intersection of two alignments, and select the intersecting alignments as the object data.

Intersection of Elm Street and Oak Street
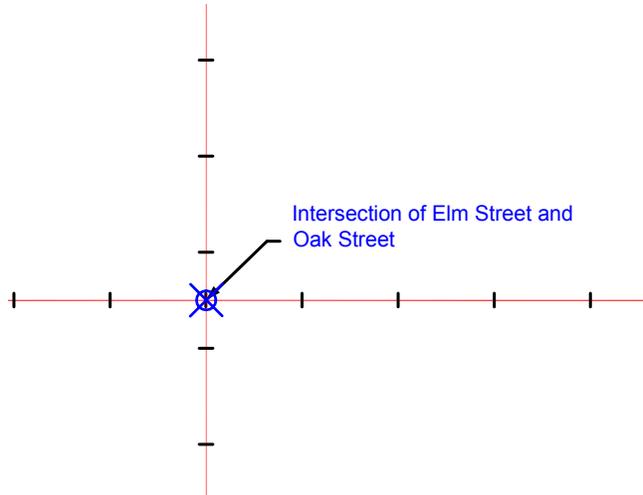
**9** If you have multiple components in the label style, be sure to specify appropriate Anchor Points and Attachment Points (page 1492) so the components do not overlap. You can use the Preview window to check the label layout.

**10** Use the other tabs on the Label Style Composer  (page 1942)to set up the label style.

---

**NOTE** Specify the default layer for Note labels on the Layers tab (page 66) of the Drawing Settings dialog box. You can also change the layer of these labels by using the AutoCADProperties palette (right-click label and select Label Properties or Properties).

---

**See also:**

■ Inserting Note Labels (page 1530)

### Quick Reference

Toolspace Shortcut Menu

  Settings tab: <drawing name> ➤ General ➤ Label Styles ➤ right-click Note ➤ New

Dialog Box

  Label Style Composer (page 1942)

## Editing Label Styles

Use the Edit Label Style command to specify a label style for an object, to create a new style, or to copy or edit an existing label style.

When you edit a label style, all labels in the drawing that reference that style are updated.

Edit an existing label style by using the Label Style Composer. You can access the Label Style Composer from Toolspace Settings tab or from the label object's context menu.

The Label Style Composer can also be accessed from the AutoCAD Properties palette by selecting Create/Edit from the available style drop-down menu. In the Label Style Control (page 1965) dialog box, select Edit Current

Selection. You can use the Style Selection pull-down menu in the AutoCAD Properties palette to bulk edit label properties for labels placed in the drawing. This is a convenient method of editing label styles without having to navigate to the Prospector Settings tab.

Notice in the following example that in addition to the Parcel Line Label Styles, the General Line Label Style is available for use when editing a Parcel Line Label.
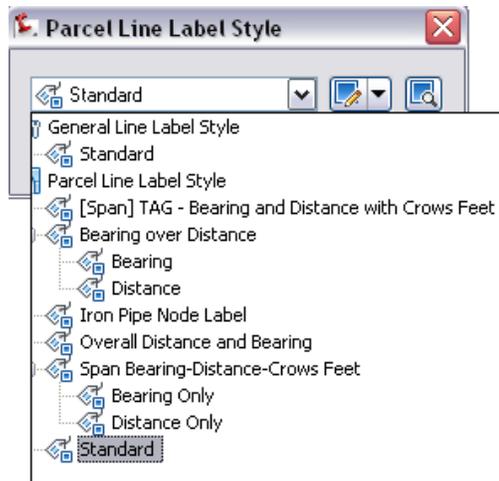


**Use right-click Edit Label Style option to access Style Selection dialog**
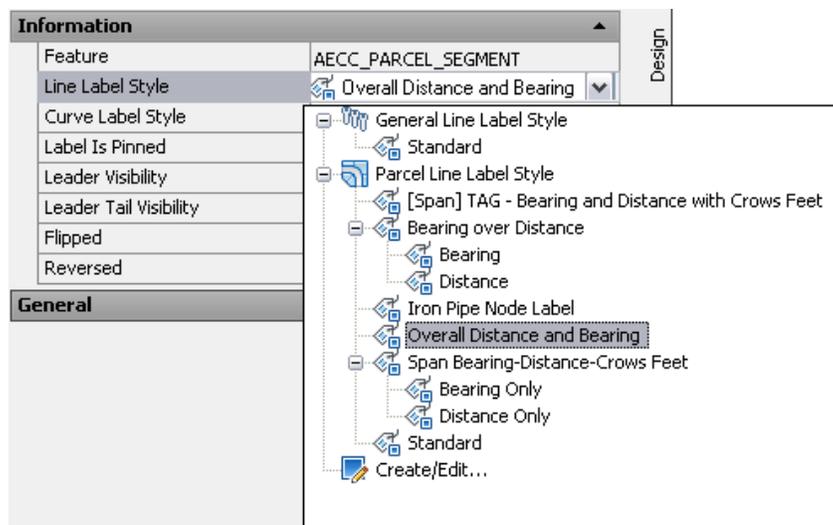


**AutoCAD Properties palette list of available label styles**

**To edit an existing label style**

1 In Toolspace, on the Settings tab, right-click the label style you want to edit and click Edit.

2 Use the Label Style Composer (page 1942) to edit the label style.

3 Click OK.

OR

1 In the drawing, select a label, right-click and select Properties or Label Properties.

2 In the Properties palette, click Line or Curve label style to display a list of available styles.

Select the Edit/Create option to access the Label Style Control (page 1965) dialog box with which you can create a new style, copy the currently selected style, create a child style, or edit the currently selected style with the Label Style Composer.

OR

1   In the drawing, select the label to edit.

2   Right-click and select Edit Label Style.

3   In the Label Style Control (page 1965) dialog box, specify a label style, create a new style, or copy or edit an existing style. Press  to access the Label Style Composer.

### Quick Reference

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Label Properties drop-down ➤ Edit Label Style 

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Modifying Label Style Content

Label content contains information that is linked directly to an object and consists of text, symbols, blocks, lines, ticks, and direction arrows. Set up this content by using the Label Style Composer.

**To modify the content of a label style**

1   In the Settings tree, right-click an existing label style and click Edit.

> **TIP** Create a new drawing using one of the default AutoCAD Civil drawing templates, which contain a set of pre-defined label styles.

2   In the Label Style Composer dialog box, click the Layout tab.

3   Click the Component Name list to see which components have been defined for the label style. The components control the label content.

4   To modify the content of a label component, select its name in the Component Name list.

5   Under Text, click in the Value column of the Contents row to display the Browse button , as shown in the following illustration:



6   Click  to display the Text Component Editor, where you can change the label properties. For more information, see Adding Text Components to Labels (page 1503).

**7**  You can also edit the static text in the label. For more information, see Editing Text in the Text Component Editor (page 1507).

# Setting Up Label Styles To Be Used as Tags

If a label style supports the use of tags and tables, there are a few unique properties you can manage in the style by changing settings in the Label Style Composer.

■  **Display Mode**. Controls whether the labels are currently displayed as labels or tags. This switches to tag mode automatically when a table is inserted into the drawing using the style-based selection method.

■  **Table Tag Component**. A unique component that is defined on the Layout tab of the Label Style Composer and is required for labels that support tables.

■  **Label "Used In" Property**. Controls whether label style components (such as text or blocks) are used in label mode, tag mode, or both.

A single label style can contain settings for both the label mode and the tag mode. You do not have to have separate label styles set up for tags. However, it may help you manage your drawings (and prevent additional styles from being created automatically) if you set up a style with its default state set to tag mode and give the style an explicit name. For more information, see Automatically-Created Child Styles (page 1479).

**To set up a label style for tags**

**1**  Create a new label style and give the style a unique name. For more information, see Creating and Editing Label Styles (page 1477).

**2**  With the new style open in the Label Style Composer dialog box, click the General tab.

**3**  Under Label, set the Display Mode to Tag. This sets the default state of this label style to tag mode. Then, when you label an object with this style, tags are created rather than full labels. For more information, see Displaying Labels as Tags (page 1487).

**4**  Click the Layout tab and select the Table Tag component in the Component Name list. This component is required for labels that support tables. You cannot delete or change the name of this component.

**5**  Under General, set the value of Used In property. For Table Tag components, the tag can be displayed in tag mode or in label and tag modes. For more information, see Layout Tab (Label Style Composer Dialog Box) (page 1944).

**6**  Under Text, click the Value column for Contents and then click ⊡ to display the Text Component Editor.

**7**  Note the property listed in the Properties list. Table tags support only the Segment Number property field. For more information, see Property Fields (page 1507).

**8**  In the Text Component Editor, edit the static text as needed. You can change "C" to "Curve" for example. Do not alter the property field itself. For more information, see Editing Text in the Text Component Editor (page 1507).

**9**  You can click the Format tab to change the tag justification or color, for example. For more information, see Formatting Text (page 1505).

**10**  Click OK in the Text Editor window and the Label Style Composer to close them.

**11**  Use the new label style for tagging objects, and then insert an object table into the drawing. For more information, see Adding Labels to Drawings (page 1522).

# Managing General Properties for Label Styles

Use the General tab of the Label Style Composer dialog box to define label style layer, visibility, display mode, and text style. The General properties also include behavior rules regarding how a label is oriented to an object, and whether it is plan-readable.

# Specifying Label Text Style

Each label style has one AutoCAD text style associated with it.

**To specify the text style to use for a label style**

1   In the Label Style Composer dialog box, click the General (page 1942) tab.

2   Under Label, click the Value column for the Text Style property, and then click ![...] .

3   In the Select Text Style dialog box, select a text style.

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Displaying Label Properties with the List Command

Use the AutoCAD List command to display label property information.

**To display the AutoCAD Text Window**

1   In the drawing, select a label.

2   In the command line, enter List.

3   The AutoCAD Text Window dialog box displays the label type, entity type, and layer.

Group type labels display "GROUP" at the end of the entity type, as well as the parent object and the default label style. For example, **AECC_ALIGNMENT_STATION_LABEL_GROUP Layer:** "**0**". In this case, the entity type is alignment station. The label type is group, and the layer is set to zero.

4   Single type labels differ from group labels as they do not display "GROUP" at the end of the entity type listing, and the label style is displayed.

> **NOTE** The alignment label type vertical geometry point label does not include "GROUP" at the end of the entity type listing. It is called: AECC_ALIGNMENT_VAGEOMPOINT_LABELING.

Command Reference

 List

Dialog Box

 AutoCAD Text Window

# Hiding and Displaying Labels in the Drawing

The label style Visibility setting controls whether the labels are visible in the drawing. You can use this setting to quickly hide and display all labels of a particular label style.

In previous releases of AutoCAD Civil 3D, most labels were sub-entities of the parent object. If the parent object was frozen, the labels would disappear with the frozen parent object. The independent label objects are no longer sub-entities of the parent object but still behave as previously. You also have the option of controlling label visibility independently through layer manipulation (with either the label style layer references, label visibility switches, or the layer that the label object itself resides on).

### To hide or display labels in a drawing

**1** In the Label Style Composer dialog box, click the General (page 1942) tab.

**2** Under Label, change the Visibility value to True or False.

**3** Click OK.

## Quick Reference

Toolspace Shortcut Menu

 Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

 Label Style Composer (page 1942)

# Displaying Labels as Tags

Display labels as tags when you want label content to be in a table.

Tags contain an identifier number that is displayed in the first column in an associated table. The other columns of the table display the object data as defined in the table style. For more information about creating tables, see Tables (page 1555).

General line and curve label styles can be displayed as tags. Tag display mode is also supported by parcel and alignment segments labels as well as curve labels. You can specify the tag start number and increment numbers in the Table Tag Numbering Dialog Box (page 1971).

**NOTE** By setting label styles to Tag display mode before creating tables, you can avoid creating unnecessary child styles. For more information, see Automatically-Created Child Styles (page 1479).

### To display labels as tags

**1** In the Label Style Composer dialog box, click the General (page 1942) tab.

**2** Under Label, change the Display Mode value to Tag.

> **NOTE** Some label types (such as parcel area labels and alignment station labels) cannot be displayed in tag mode and therefore do not have the Display Mode property.

**3** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

## Specifying the Label Style Layer

Each label style can be assigned to its own layer. The independent label objects reside on their own layer. You can also specify reference layer within the label style.

> **NOTE** Specify the default layer for labels on the Object Layers tab (page 66) of the Drawing Settings dialog box. You can also change the layer of these labels by using the AutoCAD Properties palette (right-click label and select Label Properties or Properties), or any other object layer manipulation tool.

**To specify the label reference style layer**

**1** In the Label Style Composer dialog box, click the General (page 1942) tab.

**2** Under Label, click the Value column for Layer and then click ⊡ .

**3** In the Layer Selection dialog box, select a layer, or create a new one.

**4** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

## Changing Label Orientation

Labels are composed relative to a zero angle (the default X axis), and are placed in a drawing based on an orientation setting.

Label offsets and rotation values are measured relative to the orientation setting.

**To change the orientation of labels**

**1** In the Label Style Composer dialog box, click the General (page 1942) tab.

**2** Under Behavior, change the Orientation Reference value to the desired item.

**3** Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Using Plan-Readability

Use the plan-readability settings to make label text readable in a normal plan view.

When you select the *plan-readable* option, label text is rotated to a consistent readable angle.

You can specify the angle at which to start flipping labels. By default this angle is set to 110 degrees. That means that as soon as the angle of the text goes beyond 110 degrees, the label will flip 180 degrees to remain plan-readable.

The following illustration shows labels with Plan Readable set to False. Note the label at the bottom of the illustration:
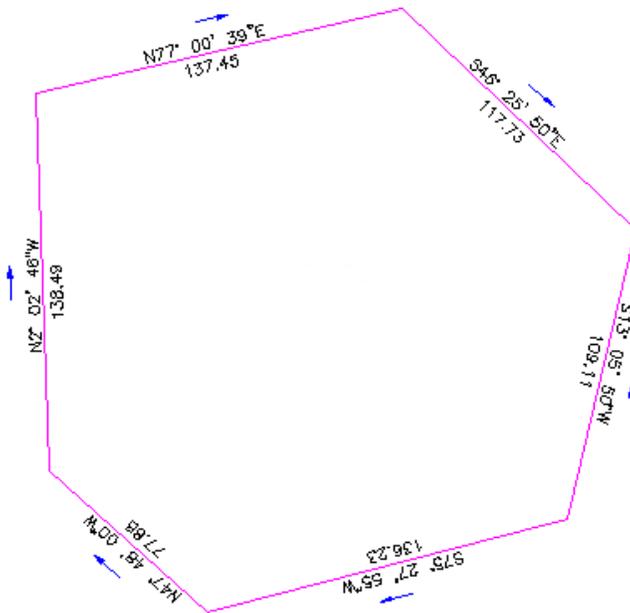


The following illustration shows labels with Plan Readable set to True:
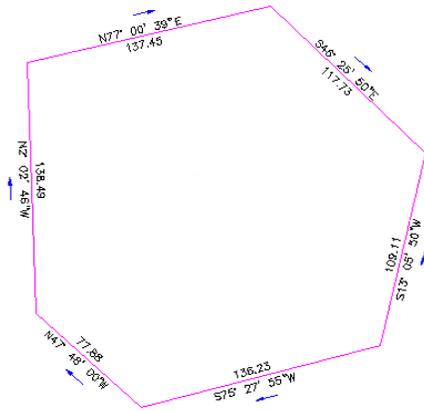
**To make a label style plan-readable**

1  In the Label Style Composer dialog box, click the General (page 1942) tab.

2  Under Plan Readability, change the Plan Readable value to True.

3  Change the Readability Bias angle to the angle at which you want the labels to start flipping. For example, an angle of 110 degrees means that label text inserted at an angle greater than 110 degrees is flipped.

> **NOTE** The labels are flipped again when they reach 180 degrees from the Readability Bias angle. For example, if the bias angle is 90, the labels are flipped at 90 degrees, and then are flipped again at 270 degrees.

4  Set the Flip Anchors With Text setting to False to flip labels so they are mirrors of the original labels. Set this setting to True if you also want to flip the label's anchor point.

> **NOTE** In most cases, it is not desirable to have this setting set to True. However, there are cases where it is needed, such as with a station label with a plus sign located always at the station tick.

5  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Changing Label Insertion

You can force labels to the top or bottom of objects or to the inside of curves.

**Top**

This setting ensures that the label is always placed on top of the object, regardless of the relationship to the zero angle of an object. The Top setting determines label placement based on the same calculation used for Plan Readability. For more information, see Using Plan-Readability (page 1489).
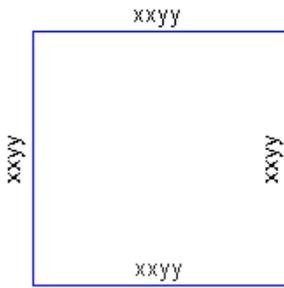
### Bottom

This setting ensures that the label is always placed beneath the object, regardless of the relationship to the zero angle of an object. The Bottom setting determines label placement based on the same calculation used for Plan Readability.
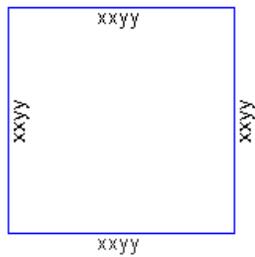


### Force to Inside of Curve

This setting ensures that the label is always placed in the inside of a curve regardless of the zero direction of the curved object.

### None

This setting places labels as they relate to the calculated zero angle of an object. It does not change label composition and, in essence, ignores any label placement rules.
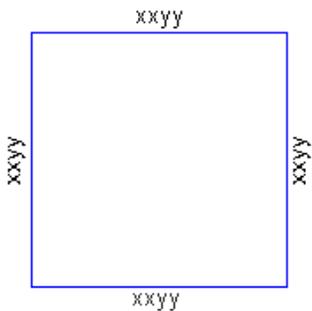


### To change label insertion

1   In the Label Style Composer dialog box, click the General (page 1942) tab.

2   Under Behavior, change the Forced Insertion value to either None, Top, or Bottom.

3   Click OK.

**To change label insertion for curves**

1   In the Label Style Composer dialog box, click the General (page 1942) tab.

2   Under Behavior, change the Force Inside Curve value to either True or False.

3   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Managing Layout Properties for Label Styles

You use the Layout tab of the Label Style Composer dialog box to control text and border settings, display properties, and content.

Layout properties are controlled independently for each label *component*. A label component can be text, a block, a line, a tick, or a direction arrow.

For example, a parcel line "Bearing Over Distance" label may have four components:

■   **Bearing**. A text component that labels the line's angle.

■   **Distance**. A text component that labels the line's distance.

■   **Table Tag**. A text component. Used only for table insertion. Required for labels that can be inserted into tables.

■   **Direction Arrow**. A direction arrow component that is positioned relative to the Bearing.

You can use the Layout properties to control each of these components separately.

**NOTE** For more information about defining new label components, see Adding Content to Labels (page 1502).

**NOTE** This section describes the Layout Properties for Text Components. Many of the same properties are available for Block, Tick, Line, and Direction Arrow components. Additional layout properties are described in Blocks (page 1513), Ticks (page 1514), Lines (page 1512), and Direction Arrows (page 1515).

## Anchor Points and Attachment Points

Label component position is defined by specifying an anchor component, an anchor point, and an attachment point. Rotation angles and X and Y offsets are applied in relation to the anchor component and anchor point.

■   **Anchor Component**. The *anchor component* is the item that the *label component* is positioned relative to. It can be the object that is labeled, or it can be another label component.

■   **Anchor Point**. The *anchor point* is the location on the anchor that is attached to the label component's *attachment point*. It can be the label location (when the anchor component is set to Feature) or it can be

one of several locations on the label component (when the anchor component is set to a specific label component).

■ **Attachment Point**. The *attachment point* is the location on the label component that is attached to the anchor point.

The following illustration shows how these points work together to position the label component (the rotated text in the illustration):
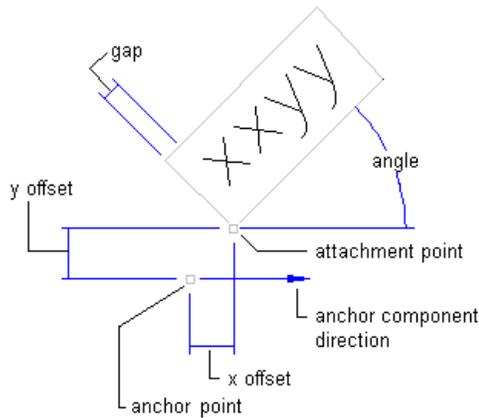


**Anchor and attachment points for text component**

The anchor is on a line (the anchor is set to Feature and the anchor point to Label Location). The attachment point on the text is set to Bottom Left. Note how the *X* and *Y* offsets, angle, and border gap all affect how the label is positioned relative to the attachment and anchor points

The following illustrations show the different points on label components that can be used either as anchor points or attachment points.
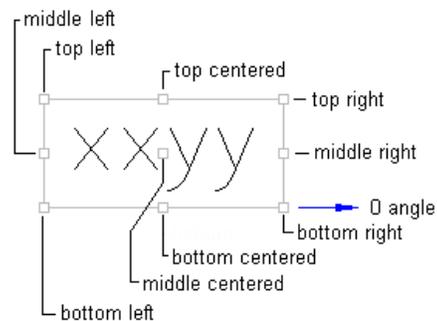


**Anchor and attachment points on text component**
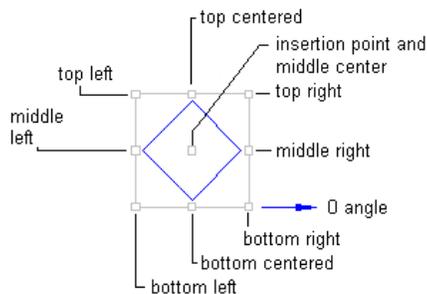


**Anchor and attachment points on a block component**

**NOTE** The location of the insertion point is defined in the block itself. The anchor and attachment points are the same on blocks and ticks.
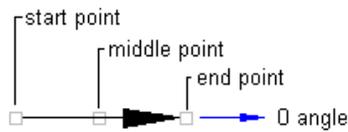


**Anchor and attachment points on a direction arrow component**

**NOTE** The anchor and attachment points are the same on direction arrows and lines.

**To specify anchor and attachment points for a label component**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

3 Under General, specify the Anchor Component value for the component.

The Anchor Component defines what the selected component should be "anchored to" — or positioned relative to—in the label.

For example, if you are setting up a Direction Arrow component for a direction label, you might want to specify the Bearing as the Anchor Component so that you can position the arrow in relation to the bearing label text.

4 Change the Anchor Point value to specify the location of the anchor point.

This value specifies precisely where on the Anchor Component you want to place the anchor.

5 Under Text, change the Attachment value to specify the location on the label component that will be attached to the anchor point.

6 Optionally, specify X and Y offsets and a Rotation Angle. For example, you may want to specify X or Y offsets from the anchor point to add some space between the components. For more information, see Offset (page 1497) and Rotation (page 1498).

**TIP** Refer to the Label Preview window when editing the label layout properties. This preview dynamically shows the effect of each of these settings.

7 Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Specifying Anchor Points

You can attach labels to specified geometry point locations.

Line and curve label anchors allow you to create much more graphic type label styles. For example, you can display crow's feet to indicate where segments start and end.

**NOTE** Anchor points do not support the spanning option for parcel segment labels.

These anchors apply to the following segment labels: parcel lines and curve Label, general line and curve labels, alignment line, and curve and spiral labels.

There are four line label anchor points that you can use to attach labels to: start, end, middle points and label location. The label location anchor can be dragged anywhere along the line.
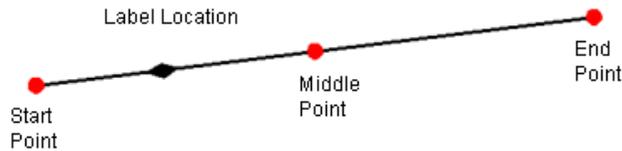


**Line label anchor points**

For spiral labels, in addition to the label location, start, end, and middle points anchors, there is a Point of Intersection anchor.
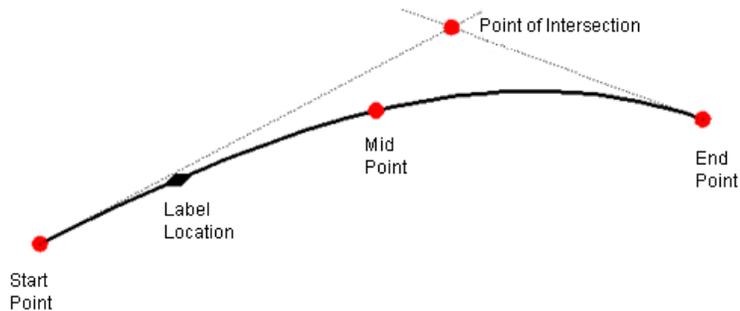


**Spiral label anchor points**

With curve label anchors, you can attach a label to the center point of the arc.
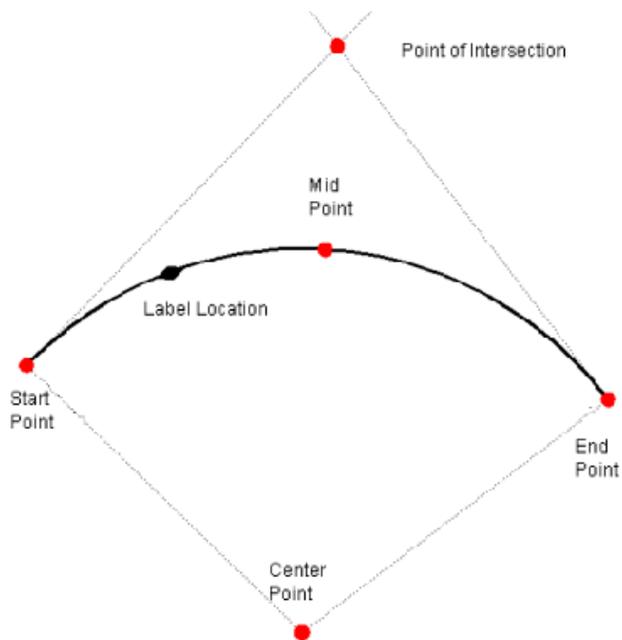
**Curve label anchor points**

**To specify label anchor point**

1  In the Label Style Composer dialog box, click the Layout tab.

2  Select an anchor point from the Anchor Point list.

3  Click Apply and then OK.

# Background Mask

Use a background mask for Text and Referenced Text components to mask the background of a label component.
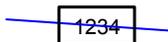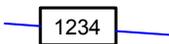


The size and shape of the mask is configured using the border settings. For example, if you specify a rectangular border with a gap of 0.1, the mask respects those settings, even if the border visibility is set to False.

You can use the AutoCAD Properties palette to include a background mask for surface contour labels

**To include a background mask**

1  In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2  Select a Text or Referenced Text component from the Component Name list.

**3** Under Border, change the Background Mask property to True.

**4** Configure the other Borders (page 1502) settings to adjust the mask and border size and shape.

**5** Adjust the label component Draw Order (page 1517) as needed.
Any label component that is drawn before the mask may be covered if it is within the mask boundary.

**To set the background mask setting for surface contour labels in AutoCAD Properties**

**1** In the drawing, right-click a surface contour label and select Properties.

**2** In the AutoCAD Properties palette, under Labels  change the Masking property to From Label Style.
This sets the contour label to the label background mask setting from the assigned label style.

OR

**3** Under Labels, change the Masking property to Contour Line Only.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Offset

Specify X and Y offsets to define the distance between the anchor point of the anchor component and the attachment point of the label component.
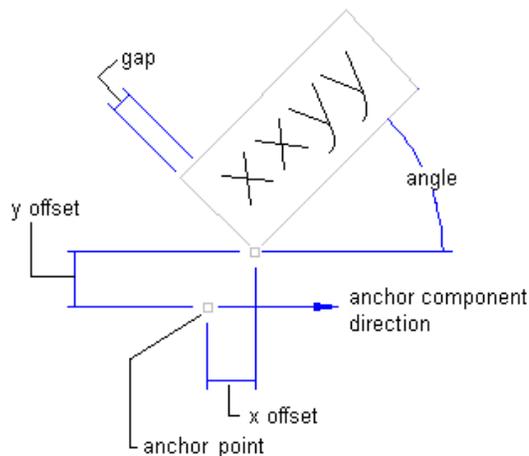


**X and Y offsets in a text component**

Offsets are dependent on the Orientation Reference of the label style:

■ Label Orientation Reference set to Object. X and Y offset values are relative to the X and Y coordinate system as defined by the anchor component direction. In the preceding illustration, the anchor component direction is zero (0) degrees, so X, and Y offsets are measured relative to zero (X direction = 0° and Y

direction = 90°). If the anchor component direction changed to 45°, then the X direction is measured at 45°, and the Y direction is measured at 135°.

- Label Orientation Reference set to World. X and Y offset values are relative to the world coordinate system (X direction = 0° and Y direction = 90°).

- Label Orientation Reference set to View. X and Y offset values are relative to the current view (X direction = bottom of screen and Y direction = 90° from bottom of screen).
  For more information about these settings, see General Tab (Label Style Composer Dialog Box) (page 1942).

**To define offset values**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

3 Change the X and Y offset values. If the label component type is a line, you must define X, Y start and endpoints.

4 Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Rotation

Specify a rotation angle for a label component.

Rotation angles are dependent on the Orientation Reference of the label style:

- Label Orientation Reference set to Object. The rotation angle is measured relative to the X and Y coordinate system as defined by the anchor component direction. For example, if the anchor component direction is 45°, then the rotation angle is measured relative to 45°.

- Label Orientation Reference set to World. The rotation angle is measured relative to the world coordinate system (X direction = 0° and Y direction = 90°).

- Label Orientation Reference set to View. The rotation angle is measured relative to the current view (X direction = bottom of screen and Y direction = 90° from bottom of screen).

**To change the rotation of a label component**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

3 Change the Rotation Angle value.

4 Click OK.

### Quick Reference

# Colors

Assign colors to individual components in a label.

Use the Select Color dialog box to assign a standard AutoCAD color or use one of the following options:

■ **ByLayer**. Labels obtain their color from the label style's layer (defined on the General tab of the Label Style Composer).

■ **ByBlock**. Labels obtain their color from the parent object's AutoCAD properties. The object is first examined for any property overrides. If none are found, then the color property of the object layer is used.

#### To specify colors for label components

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

3 Click the Value column for Color, and then click ⊞.

4 In the Select Color dialog box, select a color or select ByLayer or ByBlock.

5 Click OK.

### Quick Reference

# Linetypes

Assign linetypes to individual components in a label.

Use the Select Linetype dialog box to assign a specific linetype or use one of the following options:

■ **ByLayer**. Labels obtain their linetype from the label style's layer (defined on the General tab of the Label Style Composer).

■ **ByBlock**. Labels obtain their linetype from the parent object's AutoCAD properties. The object is first examined for any property overrides. If none are found, then the linetype property of the object layer is used.

**To specify linetypes for label components**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

> **NOTE** Text components do not have a Linetype option.

3 Click the Value column for Linetype, and then click ⊡ .

4 In the Select Linetype dialog box, select a linetype or select ByLayer or ByBlock.

5 Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Lineweights

Assign lineweights to individual components in a label.

Use the Select Lineweight dialog box to select a specific lineweight or use one of the following options:

■ **ByLayer**. Labels obtain their lineweight from the label style's layer (defined on the General tab of the Label Style Composer).

■ **ByBlock**. Labels obtain their lineweight from the parent object's AutoCAD properties. The object is first examined for any property overrides. If none are found, then the lineweight property of the object layer is used.

**To specify lineweights for a label component**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Select a component from the Component Name list.

3 Click the Value column for Lineweight, and then click ⊡ .

4 In the Select Lineweight dialog box, select a lineweight, or select ByLayer or ByBlock.

5 Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Text Height

Specify text height for label text components.

**To define text height for label components**

1  In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2  Select a component from the Component Name list.

3  Under Text, change the Text Height value.

4  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Curved Text

For label styles designed to label curves, you can specify that the label text is drawn to follow the curve.

**To define curved text**

1  In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2  Select a text component from the Component Name list.

3  Under Text, change the Allow Curved Text value to True.

4  Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

## Borders

You can add rectangular, circular, or rounded rectangular borders to individual label components.
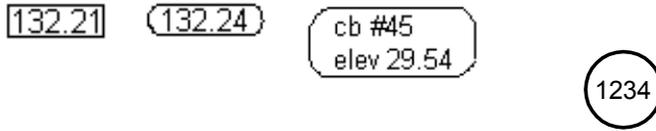


**To add borders to label components**

1   In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2   Select a component from the Component Name list.

3   Under Border, change the values for Visibility, Type, Gap, Color, Linetype, Lineweight, and Background Mask.

> **TIP** You can specify a Gap value and set the Visibility value to False to add space around the label component without actually inserting a border.

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Adding Content to Labels

You can use the Layout tab of the Label Style Composer to define *label components*, which define the content of labels. Label components include text, referenced text, blocks, direction arrows, lines, and ticks.

# Text

Use the Text Component Editor to format and define label text components.

■   To define dynamic text, select and add label *property fields*.

■   To define static text, enter text directly into the Text Component Editor window.

All text components are based on the AutoCAD text style that is defined on the General tab (page 1942) of the Label Style Composer.

## Adding Text Components to Labels

By adding text components to labels, you can create new label styles that label specific features, such as parcel area, name, and perimeter.

Text components make use of dynamic label properties that you can assign in the Text Component Editor. Each object type has several properties you can use to build the label style components.

For example, when setting up parcel line segment labels, you can choose among the following label properties:

■ Segment length

■ Segment start and end northings and eastings

■ Segment direction

■ Parcel line number

When you label a parcel using the label style, the labels are generated based on the parcel object's properties. If the parcel properties change, then the labels are updated dynamically.

There are two basic strategies you can use when adding text components to a label. If you want more precise control over the label's appearance, you can either add several properties to one text component or define a separate component for each piece of text in the label.

### Adding Multiple Properties to a Text Component

You can add multiple properties to one text component.

The advantages of this strategy include

■ Fewer text components to define

■ Fewer placement rules to define

■ The ability to place one border around the entire label

### Adding a Single Property to a Text Component

Alternately, you can add a single property per text component, and define multiple text components.

The advantages of this strategy include

■ More precise control over the placement of each piece of text

■ More property formatting options

■ The ability to turn off the visibility of one or more of the text components while leaving others visible

### To add a text component to a label

1 In the Label Style Composer dialog box, click the Layout tab.

2 Click the Down arrow in $A_+$ , and then click $A_+$ .

The component automatically receives a default name, Text.1.

3 Enter a name in the Name box.

_____

**TIP** Use a descriptive name that describes the content of the text component.

_____

4 Under Text, click the Value column for Contents and then click ⊡ :

**5** In the Text Component Editor, on the Properties (page 1962) tab, select a property from the Properties list. This list contains a set of properties applicable to each label style type.

When you select a property, Modifiers and their Values are displayed in the grid below it. These modifiers control such things as the units and precision of the labels.

**6** Change the modifier values in the Value column as needed.

**7** Click . This inserts a property field into the Text Component Editor window.

For example, if you are setting up a parcel area label, and you insert the Parcel Area property, the following property field is inserted:

<[Parcel Area(Usq_m|P2|RN|AP|Sn|OF)]>

**WARNING** You cannot edit the label property fields after you have inserted them into the Text Component Editor window. If you alter anything about the field itself, such as using the shortcut menu to change capitalization, the label style will not function correctly.

For more information about property fields, see Property Fields (page 1507).

**WARNING** To later change any of the modifier values you specified, delete the property field from the Text Component Editor window and then reinsert the property by clicking . Simply changing a modifier value and clicking OK does not update the label style.

**8** Add static text as needed in the Text Component Editor window.

For example, you might want to add "Area =" before the parcel area property field, and "square meters" after the property field.

To do this, click in the Text Component Editor window and enter the text directly. You can also insert a paragraph return after a property field and then insert additional properties to create a multi-line label.

You can use the formatting options on the Text Component Editor window shortcut menu to modify the text that you manually enter.

**9** If you want to add additional properties to the text component, select them from the Properties list and insert them, as shown in Step 7.

**10** If you want to specify different justification, fonts, or colors for the label component, click the Format (page 1963) tab in the Text Component Editor. For more information, see Formatting Text (page 1505).

**11** Click OK to close the Text Component Editor.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

## Formatting Text

Use the formatting options on the Format tab in the Text Component Editor to determine how text is displayed in a label.

Some options require you to select text in the Text Component Editor window first.

**To format label text**

1  In the Label Style Composer dialog box, click the Layout tab.

2  Select a component from the Component Name list.

3  Under Text, click the Value column for Contents, and then click ⚬⚬⚬ .

4  In the Text Component Editor dialog box, click the Format tab (page 1963).

5  Format the text as desired.

6  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

## Creating Stacked Characters

Add fractions or any other type of stacked text to label styles in the Text Component Editor.

**To create stacked characters in label styles**

1  In the Label Style Composer dialog box, click the Layout tab.

2  Select a component from the Component Name list.

3  Under Text, click the Value column for Contents, and then click ⚬⚬⚬ .

4  In the Text Component Editor dialog box, click the Format tab (page 1963).

5  In the Text Component Editor window, enter the text you want to stack separated by one of the following characters:

   ■  Slash (/) stacks text vertically, separated by a horizontal line.

   ■  Pound sign (#) stacks text diagonally, separated by a diagonal line.

   ■  Carat (^) creates a tolerance stack, which is not separated by a line.

   For example, if you want to create a fraction of 1 over 2, enter **1/2**.

6  In the Text Component Editor window, select the text that you want to stack, and then click [a/b] on the Format tab.

**NOTE** If you enter numbers separated by stack characters and then either enter a nonnumeric character or press Spacebar or Enter, the Multiline Text - AutoStack Properties dialog box is displayed. In the Multiline Text - AutoStack Properties dialog box, you can select to automatically stack numbers (not nonnumeric text) and to remove leading blanks. You can also specify whether the slash character creates a diagonal fraction or creates a vertical fraction. If you do not want to use AutoStack, click Cancel to exit the dialog box.

**7**  To unstack text, select it and click $\frac{a}{b}$ again.

**8**  To edit stacked text properties, select the stacked text, right-click, and click Properties to display the AutoCAD Stack Properties dialog box.

You can edit the upper and lower text separately. The Appearance options control the stack style, position, and text size of the stacked text.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

## Adding Special Characters

Use the Symbol option to add symbols to label text.

**To add symbols or special characters**

**1**  In the Label Style Composer dialog box, click the Layout tab.

**2**  Select a component from the Component Name list.

**3**  Under Text, click the Value column for Contents, and then click ![...] .

**4**  In the Text Component Editor dialog box, click the Format tab (page 1963).

**5**  Click in the Text Component Editor window in the location where you want to add the symbol or special character.

**6**  Click Symbol.

**7**  From the list, select a symbol, such as degree, diameter, or plus/minus, or a non-breaking space.

The symbol is added to the label text at the cursor location.

**8**  To select a special character, click Other to display the Character Map dialog box.

**9**  Select a character, click Select, and then click Copy.

**10**  In the Text Editor window, place the pointer in the location where you want to add the special character.

**11**  Paste the copied character by either right-clicking and clicking Paste or pressing Ctrl+V.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

## Editing Text in the Text Component Editor

Use a shortcut menu in the Text Component Editor window to edit the format of text content you have manually entered.

**WARNING** You cannot edit the label property fields. If you alter anything about the field itself, such as using the shortcut menu to change capitalization, the label style will not function correctly.

Right-click in the Text Component Editor window to display the shortcut menu.

The lower part of the shortcut menu contains specific formatting options. If you have selected stacked text, then options that are specific to stacked text are added to the menu.

■ **Select All**. Selects all text in the editor window.

■ **Change Case**
   ■ **UPPERCASE**. Changes all selected text to uppercase characters.

   ■ **lowercase**. Changes all selected text to lowercase characters.

■ **Remove Formatting**. Removes any formatting, such as underline, bold, and italic.

■ **Combine Paragraphs**. Removes all line returns.

■ **Stack**. Stacks selected text. Available only when text separated by stack characters is selected. For more information, see Creating Stacked Characters (page 1505).

■ **Unstack**. Unstacks text. Available only when stacked text is selected.

■ **Properties**. Displays the AutoCAD Stack Properties dialog box. Available only when stacked text is selected.

# Property Fields

Use property fields to control the label content. You add property fields to label text components to act as placeholders for the label or tag content.

## Adding Property Fields to Label Text Components

Insert property fields in text components to add content that is linked directly to drawing components.

Label text generated by most property fields is dynamic and is updated whenever changes are made to a drawing.

**NOTE** You cannot edit the property fields in the edit pane of the Text Component Editor window. You can only add or delete a property field or add static text.

As you view the property field in the edit pane of the Text Component Editor, the characters are defined and separated by the following symbols:

| Character | Description |
|---|---|
| <[ | Begin property field |
| ( | Begin modifier list |
| \| | Modifier separator |
| ) | End modifier list |
| ]> | End property field |

For example, the property field format may look like this:

```
<[Attribute(Unit|Precision|Rounding|Sign|Capitalization|Format|Direction|Output)]>
```

Property fields attributes are formatted with predefined modifiers, such as measurement units (acres, feet, and meters), precision, and rounding.

For more information about the modifiers for feature property fields in AutoCAD Civil, see Property Field Modifiers (page 1509).

**To add property fields to label text**

1  In the Label Style Composer dialog box, click the Layout tab.

2  Select a component from the Component Name list, or create a new text component.

3  Under Text, click the Value column for Contents, and then click ⋯ .

4  In the Text Component Editor dialog box, click the Properties tab (page 1962).

5  From the Properties list, select a property.

> **NOTE**  If you have defined user-defined property classifications, these are displayed in the Properties list for you to select and add to certain label styles. For more information, see User-Defined Property Classifications (page 416).

The modifier values are displayed in the grid below the property.

> **NOTE**  Because certain label style types have many possible properties, an additional filtering feature is provided for these types so you can restrict the list. Click to display the Property Filter (page 1963) Property Filter dialog box. A filter remains active until you turn it off.

6  In the Value column, change the modifier values.

7  Click to insert the property into the text window.

8  Click OK.

### Quick Reference

## Property Field Modifiers

Most label styles associated with AutoCAD Civil objects contain property fields with modifiers. These modifiers are abbreviations for the values you specified for the label property.

The following is a key to the modifier codes:

| | | | | |
|---|---|---|---|---|
| U = | Units | | S = | Sign |
| F = | Format | | D = | Direction |
| P = | Precision | | C = | Capitalization |
| R = | Rounding | | O = | Output |
| A = | Decimal character | | | |

For example, the following is a parcel area property field:

<[Parcel Area(Usq_m|P2|RN|AP|Sn|OF)]>

The following lists each of the modifiers for this label property:

- **Usq_m**: Units = square meters

- **P2**: Precision = 2

- **RN**: Rounding = Normal

- **AP**: Decimal Character = Period

- **Sn**: Sign Negative = '-'

- **OF**: Output = Full

# Referenced Text

Use Referenced Text components in label styles to include references to other objects in a label. You can insert references to surfaces, profiles, parcels, and alignments.

Referenced Text components are supported for most object label types. See Label Objects (page 1463) for a list of label objects supported by Referenced Text.

---

**NOTE** Referenced Text is supported for most label object types in AutoCAD Civil. Label types for points, parcel face, parcel segment, corridor labels, and surface watershed do not support reference text.

---

Each Referenced Text component can refer to only one AutoCAD Civil object. For example, you can format a Referenced Text component to include an alignment name and its description, but it cannot refer to two different alignment names. To include references to two different alignments you need to include two Referenced Text components in the label style, one for each different alignment.

> **NOTE** You can format a Referenced Text component to include an expression. Note that if you have referenced an expression with a property that is not supported through the referenced text property field this results in questions marks (???) inserted into the label .

**To create a Referenced Text component**

1   In the Label Style Composer dialog box, click the Layout tab.

2   Click the Down arrow in $A_+$ , and then click $A_+$ Reference Text.

3   In the Select Type (page 1949) dialog box, select the type of object data to include in the Referenced Text component. Click OK.

    You are returned to the Label Style Composer dialog box. The type of object data you selected is listed as the Reference Text Object Type in the General section.

4   Under Text, click the Value column for Contents, and then click ⋯ .

5   In the Text Component Editor dialog box, ensure the Properties tab (page 1962) is selected.

6   From the Properties list, select a property. The list of properties available reflects the object you selected in step 2 previously.

7   In the Value column, change the modifier values as needed.

8   Click ⇨ to insert the property into the text window.

9   Add other properties or descriptive text as needed. Click OK to return to the Label Style Composer dialog box.

10  Set up the other Layout settings the Referenced Text component using the same settings that are available for text components (page 1503).

> **NOTE** If the Visibility setting is set to True for the Referenced Text component in the label style, and you do not select an object for the reference when you insert the label, question marks (???) are inserted into the label. This is because the reference text component is not yet connected to an object. You can specify an object by editing the label properties and the question marks will be replaced with the object data.

**See also:**

■   Inserting Note Labels (page 1530)

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

# Text-For-Each

Use the Text Component Editor to format and define text-for-each components for a variety of object types.

*Text-for-each* components label each object associated with a parent object (the object you are labeling). For example, you can use a text-for-each component in a structure label style to include labels for pipes associated with that structure. These components are only used in structure label styles and structure tables.

Text-for-each components are formatted just like text components, except that you select which child objects to label when you first create a text-for-each component. For example, you can add labels for all the pipes that enter or exit a structure, or only for invert-in or invert-out pipes. These label components are dynamic, so that if you remove or add a child object, the labels are updated.

The following shows an example of a structure label where the first three lines of the label contain structure data. The last three lines of the label contain data about the pipes that connect to the structure:

MH - 54 TYPE "B"

STA. 20+11.35, 6.00' RT

TOP ELEV. = 62.91

INV. IN = 47.08 (N)

INV. IN = 54.42 (SE)

INV. OUT = 47.08 (W)

You would set up this label style by including two text-for-each components. One component is for all pipes in, which lists invert in values. The other component is for all pipes out, which lists the one invert out value.

**To create a text-for-each component**

1   In the Label Style Composer dialog box, click the Layout tab.

> **NOTE** The text-for-each component type is available only for structure label styles.

2   Click the Down arrow in ![A+ icon], and then click ![icon].

3   In the Select Type (page 1949) dialog box, select the type of data to include in the text-for-each component and click OK.

4   Set up the other Layout settings the text-for-each component using the same settings that are available for text components (page 1503).

For example, if you are setting up a label for invert in pipes, you could use the Text Component Editor to add static text (INV. IN =), and also add a dynamic property field that labels the pipe elevation.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

# Lines

Add line components to a label style using one of two methods.

■ **Start Point, Length, and Angle**: Use this method to anchor the start point of the line component to the anchor component. Then, specify the length and angle. The following illustration shows a line defined using this method:
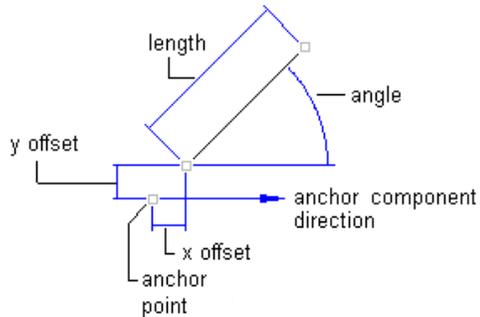


■ **Start Point and End Point**: Use this method to anchor the start and end point of the line component. The length and angle of the line are determined by the position of the anchor objects, and adjust if the anchor components are edited. The following illustration shows a line defined using this method:
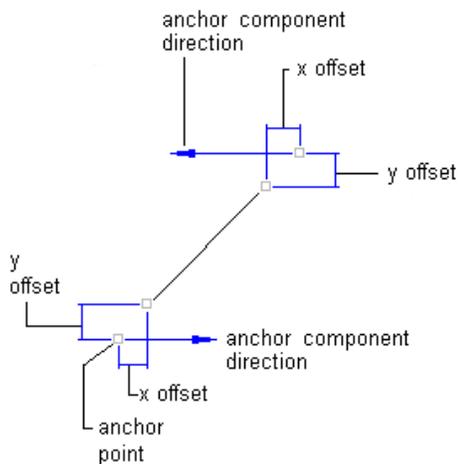


**To add lines**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Click the Down arrow in ![icon], and then click ![icon].
The component automatically receives a default name, Line.1.

3 Enter a new name in the Name box.

4 Specify the desired settings in the General and Line categories.
To define a line, either specify a start and end point anchor or define a start point anchor, a length, and a direction.
For more information, see Managing Layout Properties for Label Styles (page 1492).

5 Click OK.

### Quick Reference

Dialog Box

# Blocks

Add block components, such as a custom arrowhead or symbol, to a label style.
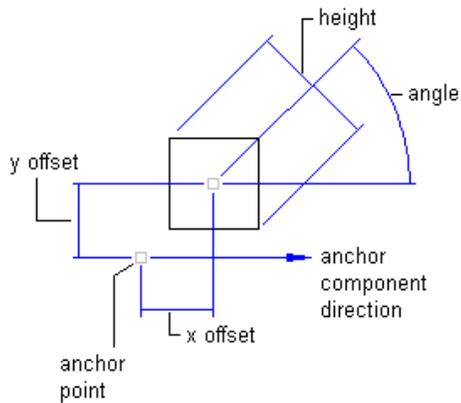


**Block component properties**

**To add blocks**

1   In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2   Click the Down arrow in  , and then click  .
    The component automatically receives a default name, Block.1.

3   Enter a new name in the Name box.

4   Under Block, click the Value column for Block Name, and then click  ⌷ .

5   In the Select A Block dialog box, select the block to use, and then click OK.

    **NOTE** The AutoCAD Civil drawing templates contain several blocks you can use.

6   Specify the desired settings in the General and Block categories.
    For more information, see Managing Layout Properties for Label Styles (page 1492).

7   Click OK.

### Quick Reference

# Ticks

You can add tick components, which are short lines inserted at a perpendicular angle to another object, to a label style. For example, ticks can be used to mark the location of a station label on an alignment.

Ticks cannot be edited with grips or moved from the object on which they are inserted. They are always anchored to the object.
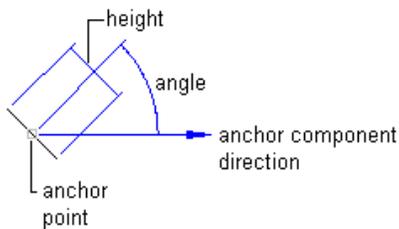


**Tick component properties**

The following illustrations show the effect of the Align With Object setting for ticks:

■ **True**. Inserts the tick at an angle relative to the object, as shown in the following illustration:
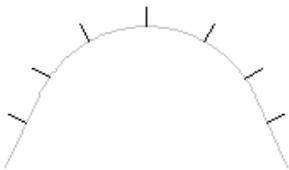


■ **False**. Inserts the tick at an angle relative to the UCS setting, as shown in the following illustration:
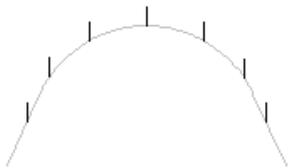


**To add ticks**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Click the Down arrow in ![icon], and then click ![icon].

The component automatically receives a default name, Tick.1.

**NOTE** This option is not available in label style types that do not support ticks.

**3**  Enter a new name in the Name box.

**4**  Under Tick, click the Value column for Block Name, and then click ⬚ .

**5**  In the Select A Block dialog box, select the block to use, and then click OK.

> **NOTE**  The AutoCAD Civil drawing templates contain several blocks you can use.

**6**  Specify the desired settings in the General and Tick categories.

For more information, see Managing Layout Properties for Label Styles (page 1492).

**7**  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Direction Arrows

Add an arrow to a label style to indicate direction.
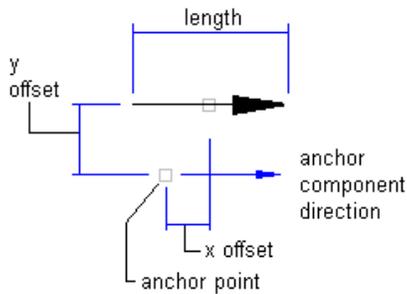


**Direction arrow properties**

The standard AutoCAD arrowheads are used in AutoCAD Civil:

| | | | | | |
|---|---|---|---|---|---|
| | architectural tick | | datum triangle filled | | oblique |
| | box | | dot | | open |
| | box filled | | dot blank | | open 30 |
| | closed blank | | dot small | | origin indicator |

| | | | | | |
|---|---|---|---|---|---|
| ∤ | closed filled | ⅃ | dot small blank | ◎ | origin indicator 2 |
| ◁ | datum triangle | ⌐ | integral | ⋁ | right angle |
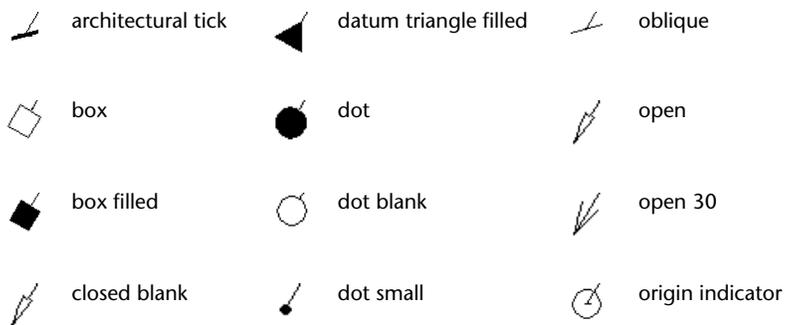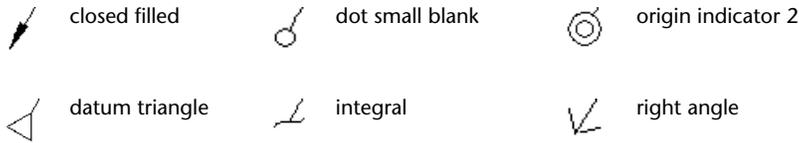
**To add direction arrows**

1 In the Label Style Composer dialog box, click the Layout (page 1944) tab.

2 Click the Down arrow in ![A+ icon] , and then click ![arrow+ icon] .

The component automatically receives a default name, Arrow.1.

> **NOTE** This option is not available in label style types that do not support direction arrows.

3 Enter a new name in the Name box.

4 Under Direction Arrow, select an Arrow Head Style.

5 Specify the desired settings in the General and Direction Arrow categories.

> **NOTE** The direction arrow length must be at least twice that of the specified arrow head size. If not, the arrow head is not displayed. For example, if you specify an arrow head size of 0.2, the arrow length must be at least 0.4. This behavior is similar to AutoCAD leaders.

**Arrow head is not displayed when
arrow length is decreased**
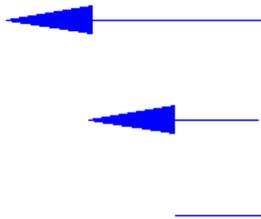


6 Click OK.

For more information, see Managing Layout Properties for Label Styles (page 1492).

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Copying Label Components

To quickly create a new label component, you can copy and edit an existing component.

**To copy a label component**

**1** In the Label Style Composer dialog box, click the Layout (page 1944) tab.

**2** Select the component you want to copy from the Component Name list.

**3** Click ✨ Copy Component.

A copy is created and named "Copy of" the component you selected.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ Right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Changing Label Component Draw Order

You can change the label component draw order to control which label components are drawn on top of other components.

This is useful if you want to use a mask in a label style to cover drawing entities and then have another component appear on top of the mask.

**To change label component draw order**

**1** In the Label Style Composer dialog box, click the Layout (page 1944) tab.

**2** Click ⬚ Draw Order.

**3** In the Component Draw Order (page 1958) dialog box, select a label component in the list and click the Up or Down arrows to move it up or down in the display order. The item at the top of the list is drawn last, so it will appear on top of all other objects.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Component Draw Order (page 1958)

# Managing Dragged Properties for Label Styles

Use the Dragged State tab of the Label Style Composer dialog box to define leader visibility and properties for dragged label text.

You can display dragged label text as you composed it originally, or as stacked text, as shown in the following illustrations:
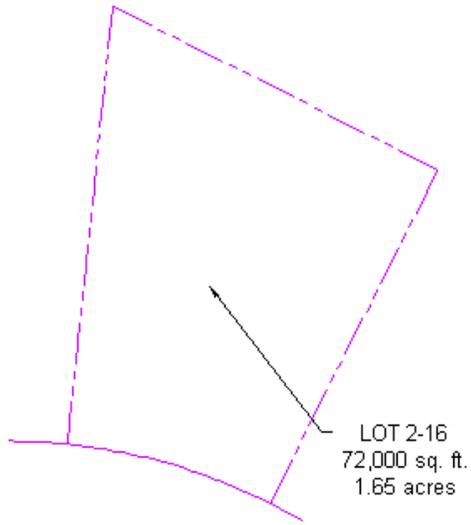
**Label "As Composed"**
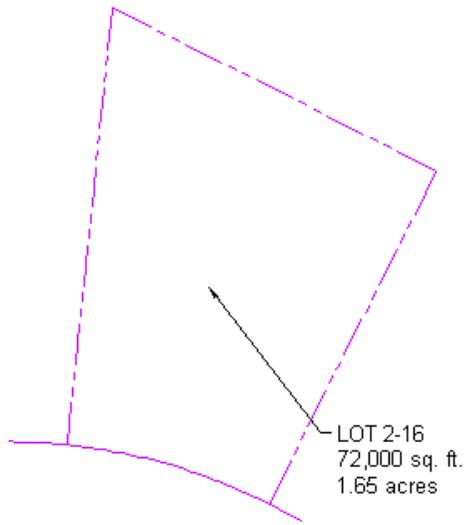


**Stacked label**

# Managing Leader Properties for Label Styles

Leaders can be displayed when you drag a label away from its default location.
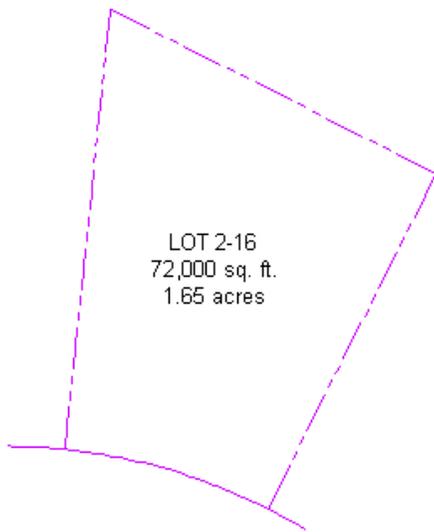
**Default label location**
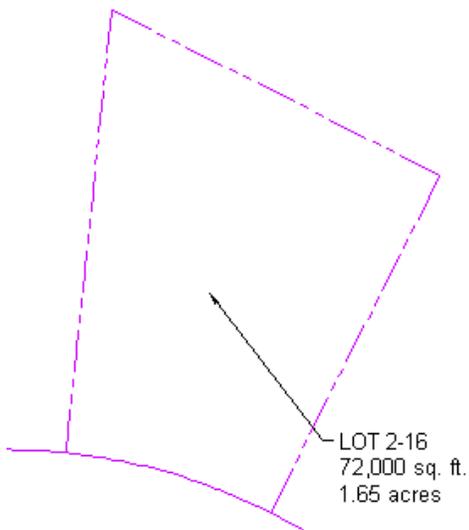


**Dragged label with leader**

A leader line remains attached to the original insertion point of the label and stretches to the label's new location. A leader can be drawn with an arrowhead and can be a straight line or a spline curve.



**Straight leader**



**Spline leader**

Text in a dragged label is stacked according to settings that you specify in the Dragged State (page 1958) tab of the Label Style Composer.

**To manage leader properties**

**1** In the Label Style Composer dialog box, click the Dragged State (page 1958) tab.

**2** Under Leader, change the Visibility value to True to display leaders when you move a label away from its original placement.

**3** Specify the leader arrow head style and size.

**4** Specify the leader type, either Straight Leader or Spline Leader.

**5** Specify color, linetype, and lineweight properties.

**To display/hide the leader**

■ Right-click a label and click Toggle Leader.

**To display/hide the leader tail**

■ Right-click a label and click Toggle Leader Tail.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Managing Leader Vertices

Leader vertices can be added, moved, or removed.

Use the leader vertex grips to add, move, and reset leader vertices. You can add as many leader segments as desired.
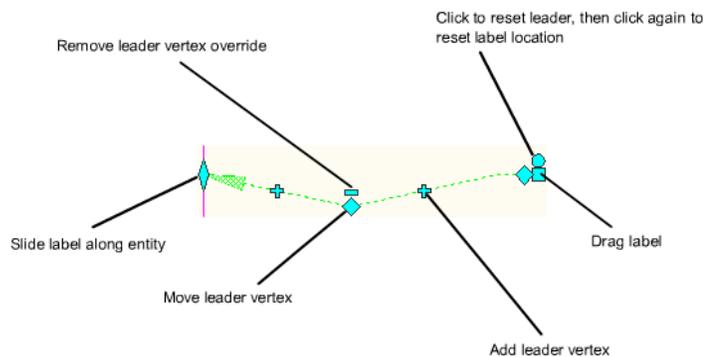


**To add a leader vertex**

**1** Click on a dragged label and select the ✚ grip.

**2** Move the grip to the new location. A vertex is created. Note that new ✚ grips display on the resulting leader lines. Use these grips to create additional vertices.

**NOTE** If the leader type in label style dragged state is set to spline, an AutoCAD spline fit is made passing through the vertex points.

**To move a leader vertex**

1 Click on a dragged label that has at least two leader lines.

2 Select the grip and select a new vertex location.

**To remove a leader vertex**

1 Click on a dragged label that has at least two leader lines.

2 Select the grip and the adjacent vertex is reset.

**To reset leaders**

1 Click on a dragged label that has at least two leader lines.

2 Select the grip and the leader line is reset to its original state.

**NOTE** Click the grip a second time to reset the dragged label to its undragged state.

# Managing How Label Text Appears when Dragged

Dragged label text can be displayed as composed, which means that it is displayed the same way when dragged as when it is originally inserted, or it can be stacked. When stacked, all blocks, lines, and direction arrows are removed.

You can add borders around stacked label text, and specify how the text is justified. Label text is left-justified when the leader is on the left side of the text and is right-justified when the leader is on the right, as shown in the following illustration.

LOT 2-16
72,000 sq. ft.
1.65 acres

LOT 2-16
72,000 sq. ft.
1.65 acres

However, you can set the label text to be always left-justified regardless of the location of the leader, as shown in the following illustration.

LOT 2-16
72,000 sq. ft.
1.65 acres

LOT 2-16
72,000 sq. ft.
1.65 acres

You can also define the leader attachment position, which defines where the leader appears in relation to the label content.

**NOTE** If you set the display of a dragged label component to As Composed, then all the Dragged State Components properties are unavailable for editing. To control leader placement for "as composed" labels, edit the text attachment, anchor point, and offset settings on the Layout tab.

**To control how label text appears when dragged**

1 In the Label Style Composer dialog box, click the Dragged State (page 1958) tab.

2 Under Dragged State Components, change the Display value to either As Composed or Stacked Text.

> **NOTE** When you use the As Composed setting, you may need to adjust the text attachment point on the Layout tab to prevent the leader from crossing the text. For more information, see Anchor Points and Attachment Points (page 1492).

3 If you select Stacked Text, specify the border and leader values, including the Leader Justification value:

- **True**. Text is right-justified when the leader is on the left side of the text and is left-justified when the leader is on the right.

- **False**. Text is always left-justified regardless of the location of the leader.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)

# Inserting and Managing Labels in Drawings

You can add labels to drawing objects, change label properties, flip and pin labels, and renumber label tags.

# Adding Labels to Drawings

You usually add labels to drawings when you create objects such as points, alignments, or parcels. You can also add labels after you create the objects.

- You specify the label styles to use when creating objects, and these labels are created automatically. For more information, see the Help about creating individual features.

- You can also insert labels using the Add Labels (page 1970) dialog box, and by using other feature-specific commands, such as the surface contour label command.

- You can also insert labels using the Annotate tab. This tab provides an efficient work environment for when you are ready to label objects that you have created. This tab provides quick access to all AutoCAD Civil feature-specific commands. Before using this tab, set your default label settings for best results as you are able to quickly label objects in rapid succession.

The labels that are created when you create objects can be deleted. Specifically, you can erase a whole group of labels and restore them by resetting the label group. You can also hide a single label and restore its visibility by resetting the label group. You can also explode labels and delete the labels that you manually insert.

**To manually add labels to drawings**

1 Click Annotate tab ➤ Labels & Tables panel ➤ Add Lables Drop-down ➤ Add <feature-specific label> .

2 In the Add Labels (page 1970) dialog box, select the desired label type and style.

**3** Click Add and select the label location in the drawing.

### Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ <Feature ➤ Add <Feature> Labels

Menu

<Feature> menu ➤ Add <Feature> Labels

Dialog Box

Add Labels (page 1970)

#### To add feature specific labels to drawings

➤    Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels drop-down ➤ Add <feature-specific label> .

OR

**1** Click <feature> tab  ➤  Labels & Tables panel  ➤ <command>.

**2** Select the object to add the label to.

# Labeling Lines and Arcs

You can label lines, arcs, 2D polylines, feature lines, parcels, and alignments with AutoCAD Civil label styles. The general line label styles are available for lines, arc, 2D polylines, feature lines, lot lines, and alignments. You can also convert these general line labels to tags.

When creating or editing line labels, you can select both parcel label/alignment label styles as well as general line and curve label styles. The commands to add single and multiple label types support each of the three object types (line/curve, parcel, and alignment).

#### To label lines, arcs, polylines, feature lines, parcel, and alignments with general line label styles

**1** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line and Curve  ➤  Add Line and Curve Labels ✎ .

**2** In the Add Labels (page 1970) dialog box, under Feature, select Line and Curve.

**3** Under Label Type, select Single Segment or Multiple Segment. Multiple Segment labels all segments on a selected polyline or feature line.

**4** Select the Line and Curve label styles to use. For more information, see Creating Label Styles for Lines, Curves, and Polylines (page 1479).

Note that you can select parcel, alignment, and general line and curve label styles to annotate lines.

**5** If the label styles use Referenced Text (page 2481), specify the Reference Text Object Prompt Method to use.

**6** Click Add. Then select the object to label.

**Quick Reference**

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Line and Curve ➤ Add Line and Curve Labels

Menu

General menu ➤ Add Labels

Command Line

AddSegmentLabel
AddSegmentLabels (for multiple segments)

Dialog Box

Add Labels (page 1970)

# Labeling the Overall Length of Parcels

AutoCAD Civil contains label properties that you can use to label the overall length on the outside of contiguous parcels.

**To label the overall length of parcel segments**

1 Create a new parcel line label style or open an existing style to edit.

> **NOTE** You can also set up parcel curve label styles to span multiple segments.

2 On the Layout tab of the Label Style Composer, select or create the label component that you want to span multiple segments.

3 In the Properties list, under General, change the Span Outside Segments property to True.

Specify this setting for each component that you want to span multiple segments.

> **NOTE** The Span Outside Segments option works only on outside segments. If you have this option specified for a component, but it is positioned so that it appears inside a parcel, it will label the individual segment, not the spanned segment.

**Even if a label is set to Span Outside Segments, it must be positioned outside the parcel to label the span**
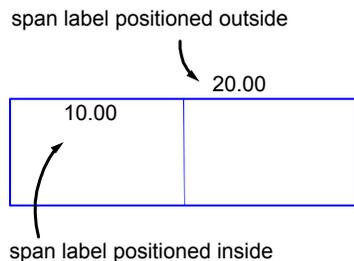
**4** Make sure that the components are all set up to display on the outside of the parcel line.

- On the Layout tab, adjust the Anchor Component, Attachment position, and Y Offset values as needed.
  For example, if the Anchor Component is Feature, set the Attachment to Bottom Center adjust the Y Offset to a positive number so the label will appear above the feature.

- On the General tab, set the Orientation Reference setting to Object and the Forced Insertion setting to None.
  The None setting forces the label component to the outside of most segments. However, on some segments, you may need to use the Flip Label command to flip the label to the outside.
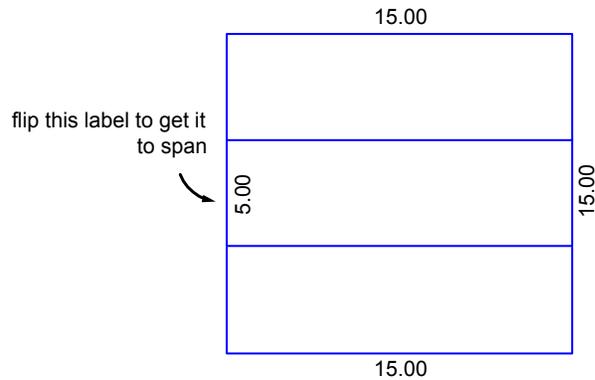


**5** To insert the labels, use the Parcels ➤ Add Labels command, select Single Segment as the label type, and select the label style that you set up to span.

The span label is inserted at the location you click, and the values contain the overall data for the contiguous parcels.

The parcel segment to label with the span must be one unbroken entity. For example, in the tutorial drawing Parcel-3.dwg, you can see that the top of the parcel group is one segment, so a span label can be successfully applied.
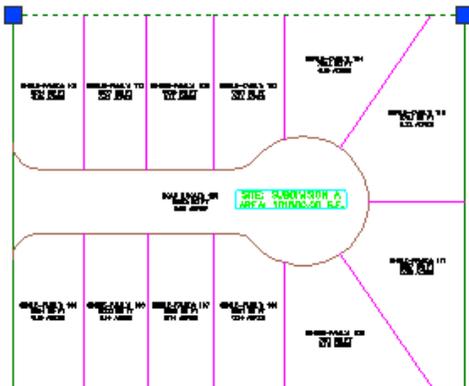


However, the left side of the parcel group is broken up into three segments, as shown below. The span label will not label all three. If you want to label the overall distance of that side, you would need to redraw the left side as one parcel entity.
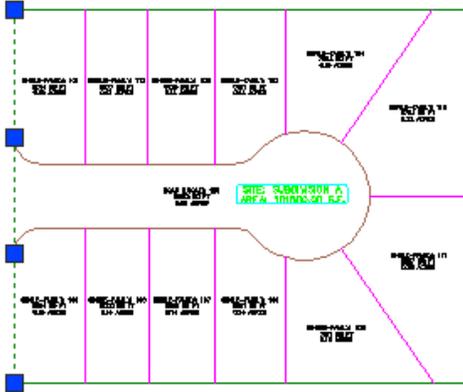
# Setting Up Profile and Section View Labels

Because profile views and section views can contain more than one profile or section, certain profile and section labels require that you specify which profile or section data to use for the labels.

Specify which profile or section data to use by modifying the label properties after you add the label to the drawing. The label styles that use this feature are

■ Profile view Station Elevation labels

■ Section view Station Offset labels

The following workflow describes how to set up the profile view labels, and the same concepts can be applied to setting up the section view labels.

**To create profile view labels that contain data for multiple profiles**

1   Create a profile view that contains multiple profiles.

   **NOTE** The *Profile-3.dwg* file in the *\AutoCAD Civil 2010\Help\Civil Tutorials\drawings* folder contains a profile view you can use for experimenting.

2   Create a new profile view Station Elevation label style, or open an existing profile view Station Elevation label style for editing. For more information, see Creating and Editing Label Styles (page 1477).

3   With the style open in the Label Style Composer dialog box, click the Layout tab.

4   Verify that the Station Elevation Data component is selected in the Component Name list.

5   Under Text, click the Value column for Contents, and then click ⋯ to display the Text Component Editor.

6   Expand the Properties list. Note the following properties you can specify:
   ■ Profile 1 Name

   ■ Profile 1 Elevation

   ■ Profile 1 Elevation Difference

   ■ Profile 2 Name

   ■ Profile 2 Elevation

   ■ Profile 2 Elevation Difference

- Profile 1 Elevation minus Profile 2 Elevation

- Profile 2 Elevation minus Profile 1 Elevation

Because there may be multiple profiles in a profile view, you can assign any of those profiles to be "Profile 1" or "Profile 2." After you add these property fields to the label style and label the profile view, you can edit the label properties to specify which profiles you want to use.

7   In the Text Component Editor window, place your cursor at the end of the existing property fields and press Enter to create a new row.

8   In the Properties list, select a Profile 1 or Profile 2 property to add, and then click ⇨ to insert the property field. For more information, see Adding Text Components to Labels (page 1503).

For example, you could insert the "Profile 1 Name" property, enter a colon, and then insert the "Profile 1 Elevation" property, enter a paragraph return, and then insert similar properties for Profile 2.

9   Click OK to save your changes and close the dialog boxes.

10   Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Profile View ➤ Add Profile View Labels 🏷 .

11   In the Label Type list, select Station Elevation.

12   In the Station Elevation Label Style list, select the new style you created or the style you edited.

13   Click Add, and then select the profile view to label.

14   Specify the station and the elevation, and then press Enter to end the command.

The label is inserted with question marks (???) as placeholders for the profile 1 and profile 2 data.

15   Select the label, right-click, and click Label Properties.

16   In the Label Properties dialog box, click the Profile 1 Object column, and then click ⊡ .

17   In the Profile 1 Object dialog box, select the profile that you want to use as Profile 1, and then click OK.

18   Repeat Steps 14 and 15 for the Profile 2 Object.

19   Click OK to close the Label Properties dialog box.

The label is updated with the profile data.

---

**NOTE** If the label does not update immediately, enter Regen at the command line.

---

20   Use the label staggering options to avoid label collisions and to improve readability. For more information, see Staggering Profile and Section Labels (page 1527).

# Staggering Profile and Section Labels

Staggers profile and section labels for improved visibility.

Use the label height staggering options to avoid collisions in densely populated graph views in order to improve legibility. The staggering options are available through the AutoCAD Properties palette or the Properties dialog boxes.

Specify the height of the stagger lines so that label components have adequate spacing.
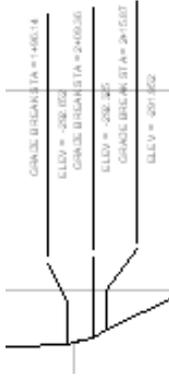
**Profile with staggered labels to avoid collisions**

**To stagger profile labels**

1   Right-click a profile object and select Edit Labels.

2   In the Profile Label Set (page 2167) dialog box, specify the Stagger Labels settings as required.

3   Specify the height of the stagger lines.

**NOTE** Layer, linetype and other AutoCAD properties of the stagger lines are determined by the leader settings in the Dragged State tab of the corresponding label style.

OR

1   Right-click a profile label and select Properties

2   In the AutoCAD Properties palette under Staggering, specify the stagger label settings as required.

# Staggering Data Band Labels

Staggers data band labels for improved visibility.

Data band labels are objects that can be manipulated like other AutoCAD Civil label objects. Use the label height staggering options to avoid label collisions in densely populated data bands. The staggering options are available through the Profile View Properties or the AutoCAD Properties palette.

**NOTE** Data band labels can also be grip edited like other label objects.
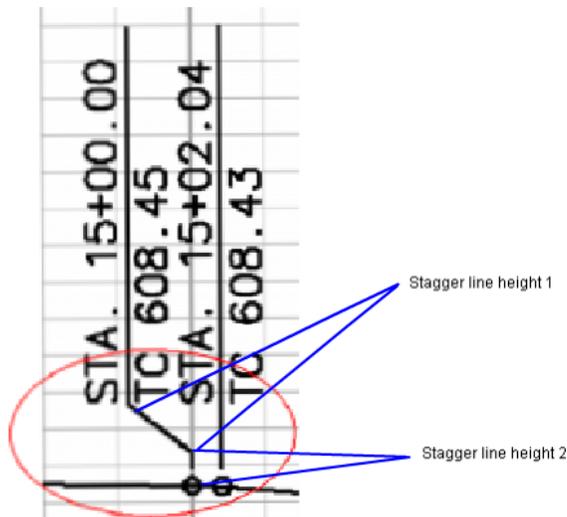
**You set the stagger label line height**

**The staggering options are available for the following label types**

- Grade breaks (profile labels)

- Major/minor stations (profile labels)

- Horizontal geometry points (profile labels)

- Major/minor offsets (section labels)

- Grade breaks (section labels)

The following workflow describes how to stagger profile view labels, and the same concepts can be applied to setting up the section view labels.

**To stagger data band labels**

1 Right-click a data band label and select Profile View Properties.

2 In the Profile View Properties dialog box, click the Bands (page 2181) tab. Specify the Stagger Labels setting as required.

3 Specify the height of the stagger lines.

OR

1 Right-click a data band label and select Properties.

2 In the AutoCAD Properties palette under Staggering, specify the stagger label settings as required.

# Editing Data Band Labels

You can edit data band label objects with the Label Style Composer.

Use the Label Style Composer to specify a label style for an band, to create a new style, or to copy or edit an existing label style.When you edit a label style, all band labels in the drawing that reference that style are updated.

**To edit data band label styles**

**1** Right-click a data band label and select Edit Band Style.

**2** In the Profile Data Band Style dialog box, click the Band Details (page 2165) tab. Select the Compose Label button.

**3** Use the Label Style Composer Dialog Box (page 1942) to edit the data band label style.

**4** Click OK.

# Inserting Note Labels

You can insert Note labels that are not anchored to a specific object.

Note labels can contain text strings as well as Referenced Text (page 2481), which refers to other objects in the drawing.

It is important to remember that although the Referenced Text components in a Note label will update if a referenced object changes, a Note label itself is not attached to a specific object. Therefore, it does not move if you move a referenced object or an object that you have snapped it to. You must manually move a Note label to reposition it.

You can edit (or override) Note labels in a drawing to change the text for unique information. That way you do not have to have a separate label style for each note you want to insert. For more information, see Overriding Label Text (page 1535).

**To insert note labels**

**1** Specify the layer for Note labels on the Layer tab (page 66) of the Drawing Settings dialog box.

**2** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Notes � .

**3** In the Add Labels (page 1970) dialog box, under Feature, select Note.

**4** Select the Note label style to use. For more information, see Creating Note Labels (page 1480).

**5** Optionally, select a Marker Style to use. Using a Marker Style places a symbol at the Note label insertion point. If you do not select a Marker Style now, you can specify one later by editing the Label Properties.

---

**NOTE** You can set up new Marker Styles from the Toolspace Settings tab. Expand the General collection ➤ Multipurpose Styles ➤ Marker Styles.

---

**6** If the label style uses Referenced Text (page 2481), specify the Reference Text Object Prompt Method to use.

**7** Click Add.

**8** Select the location in the drawing where you want to insert the note.

**9** If the label uses Referenced Text, select the object(s) to reference by doing one of the following:

■ If you specified Command Line as the prompt method: Refer to the command line prompts to select the objects.

■ If you specified Dialog as the prompt method: The Label Properties (page 1966) dialog box is displayed. Click the Value column next to the Referenced Text item (where <none> is listed). Then click ⋯ and select the object from the Property Field Object dialog box.

**NOTE** If the Visibility setting is set to True for the Referenced Text component in the label style, and you do not select an object for the reference when you insert the label, question marks (???) are inserted into the label. You can specify an object by editing the label properties.

**NOTE** See Referenced Text (page 1509) for a list of other label types that support Referenced Text.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Notes

Menu

General menu ➤ Add Labels

Command Line

AddNoteLabel

Dialog Box

Add Labels (page 1970)

# Labeling External References (Xrefs)

You can label xref objects as you would native objects.

The process to label xref objects is the same as if you were labeling an object in the current drawing. If the xref object is modified in the source drawing, then the corresponding label updates dynamically.

Labels you add to an xref exist in the current drawing. If you detach the xref, labels created in the current drawing will be lost. Xref data is read-only in the current drawing and can only be edited in the source drawing.

**For Xref labels to be created**

1    The xref must be inserted into the current drawing at 0, 0, 0.

2    The xref must be scaled at 1,1,1.

3    The xref must not be rotated.

Objects in an xref cannot be individually context selected. Xref commands, such as detach, bind, and unload, can alter the association between a label and the object within an xref. For more information, see Xref Label Behavior (page 1532).

If a drawing containing AutoCAD Civil objects and labels is inserted as an xref into another drawing, the labels will not be altered. It is recommended that if you insert xrefs that include labels, you should set the labels on layers that can be frozen.

**NOTE** Xrefs can only be used to create parcel and alignment tables. Other xref objects are not currently supported by tables.

**To add Xref labels**

1    Click Insert tab ➤ Reference panel ➤ Attach .

**2** In the Select Reference File dialog box, select the DWG to insert into the current drawing.

**3** In the External Reference dialog box, set the Insertion Point values (X, Y, and Z) to 0.00.

**4** Set the Scale values (X, Y, and Z) to 1.00.

**5** Set the Rotation angle to 0. Press OK.

The xref object displays in the current drawing. Now you can label the xref data.

**6** Click Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Notes ✎ .

**7** Select the label style to use.

> **NOTE** Labels for View Frames and Matchlines are NOT supported through xrefs.

**8** Click OK and pick the label locations.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Labels menu ➤ Notes ✎

Menu

General menu ➤ Add Labels

Command Line

AddLabels

Dialog Box

Add Labels (page 1970)

## Xref Label Behavior

Existing Xref labels are affected by edits to the xref object.

When an xref is modified, the xref label can be affected and/or removed.

The following describes the behavior of existing xref labels if the xref is modified through xref or AutoCAD commands.

| Command | Result |
| --- | --- |
| Xref Unload | While unloaded, the labels temporarily disappear, but still exist. When the xref is reloaded, the labels reappear. |
| Broken (Xref not found) | This occurs when an xRef file is renamed, deleted or path to it changed. While not found, the labels temporarily disappear, but still exist. When the xref is reloaded, the labels reappear. |
| Xref Detach (or erase) | The labels are erased and cannot be restored. |

| Command | Result |
|---|---|
| Xref Reload | The labels updated accordingly to the xref. If the xref has been modified, the label updates. If the xref has been removed, the label is erased. |
| Bind Xref | The label is erased. |
| Explode Xref | The label is erased. |
| AutoCAD command that alters the size or position of the xref. Such as move, rotate, or stretch. | Labels are converted to a single line of text "???", at the new insertion point. When the xref is altered back to meet the insertion requirements listed previously, the labels regenerate and appear as designed. When an xref is moved, the label displays xref moved. |

You cannot select a pick-sensitive context (right-click) menu for an object within an xref. Some labeling commands are available in <object> feature menus so they can be used with xref objects. Commands for editing alignment station labels (page 973), profile labels (page 1033), sample line (page 1136), and section labels (page 1137) are now available in their respective label menus on the Annotate tab.

# Identifying and Selecting Xrefs

Xrefs are easy to identify when selecting reference text objects for label styles or objects for inclusion in tables.

The name of the xref object uses this convention: **<XRef DWG Name>|<Object name in XRef>**. Note that the XRef DWG Name does not include the file extension or path. This is the same convention that AutoCAD uses for xref layer names.

# Modifying Labels in a Drawing

You can modify label object properties by right-clicking a label and selecting Label Properties or Properties to open the AutoCAD Properties palette. Use Ctrl+click to select individual labels within a group for editing. You can also grip edit label components, flip, pin, and reset labels.

## Changing Group Label Object Properties

Turn off the visibility of a group of labels, change the label style, and specify other properties depending on the type of label selected.

**To change group label properties**

1   Select a label in a drawing (the entire group is selected), right-click, and select Label Properties or Properties.

2   In the Properties palette, change the desired properties.

3   Click OK.

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Edit Label Group 

Object Shortcut Menu

Right-click label ➤ Label Properties or Properties

## Changing Single Label (single label within a group) Properties

To turn off the visibility of the label, change the label style, and specify other properties depending on the type of label selected, you can select sub-entities (single labels) within a group label using Ctrl+click.

**To change sub-entities (single labels) within a group label**

 **1**  Press Ctrl+click to select labels in a drawing.

 **2**  Right-click and select Label Properties or Properties.

 **3**  In the Properties palette, change the desired properties.

 **4**  Click OK.

OR

 **1**  In the drawing, select the label to edit.

 **2**  Right-click and select Edit Label Style.

 **3**  In the Label Style Control (page 1965) box, specify a label style, create a new style, or copy or edit an existing style. Press  to access the Label Style Composer.

### Quick Reference

Object Shortcut Menu

Press Ctrl and right-click label ➤ Label Properties or Properties

## Changing Non-Object Type Label Properties

Properties for points labels, all corridor labels, and surface watershed labels are modified with the Label Properties dialog box. Turn off the visibility of the label, change the label style, and specify other properties depending on the type of label selected.

**To change label properties**

 **1**  Select a label in a drawing, right-click, and click Label Properties or Properties.

 **2**  In the Label Properties (page 1966) dialog box, change the desired properties.

 **3**  Click OK.

### Quick Reference

### Quick Reference

## Overriding Label Text

Use the Edit Label Text command to override content of individual labels.

For certain label object types, you can override the content of individual labels. This override is useful if you want to add additional text specific to a certain label.

If you have edited the label text, this allows you to clear those edits. The label will only display what is in the style.

Overrides are supported for the following label types:

- Line
- Curve
- Note
- Profile View Station Elevation
- Surface Spot Elevation
- Section View Offset Elevation
- Segment
- Surface Slope
- Surface Contour - Major, Minor, User
- Alignment Major Stations
- Alignment Minor Stations
- Alignment Geometry Point
- Alignment Station Equation
- Alignment Design Speed
- Alignment - Profile Geometry Point
- Alignment Station Offset - Fixed Point
- Alignment Station Offset
- Alignment Line
- Alignment Curve
- Alignment Spiral

- Alignment Tangent Intersect

- Profile Major Stations

- Profile Minor Station

- Profile Horizontal Geometry

- Profile Grade Breaks

- Profile Line

- Profile Sag Curves

- Profile Crest Curves

- Profile View Depth

- Profile View - Station Elevation

- Sample Line

- Section Major Offset

- Section Minor Offset

- Section Grade Break

- Section Segment

- Section View Grade

- Network Pipe in plan

- Network Pipe in profile

- Network Pipe in section

- Network Structure in plan

- Network Structure in profile

- Network Structure in section

- Network Pipe - Spanning pipe

- Plan Production - View Frame

After you have overridden a label component, any changes you make to the text content in the label style are not applied until you clear the label text overrides. Changes you make to the non-text content in the label style (such as border, visibility, or color) are still applied to the overridden label.

**To override label text**

1  Select the label in the drawing. Right-click and click Edit Label Text.

2  The Text Component Editor (page 1961) is displayed.

3  Edit the label component as needed. Click OK.

**To clear label overrides**

1  Right-click the label and click Clear Label Text Override.

If multiple text components have been overridden for the label, you are prompted to select the text component to clear.

**2** Do one of the following:

**To clear all label overrides**

**1** Select the label(s) that contain the overrides.

**2** Right-click and click Clear All Label Text Overrides.

## Quick Reference

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Clear Label Text ✎

Command Line

EditLabelText

## Quick Reference

Object Shortcut Menu

Right-click label ➤ Edit Label Text

Command Line

EditLabelText

# Moving Labels in a Drawing

Use grips to move labels. Each label in a drawing contains a single, primary grip that can be used to drag a label from its position.

The Dragged State settings within the label style control how labels appear when dragged. If the Stacked Text option is selected, text components in dragged labels are stacked vertically based on the order in which the text components were created in the label style. Dragged labels can display a leader pointing from the text to the label's original location. For more information about stacked text, see Managing How Label Text Appears when Dragged (page 1521).

**To move a label in a drawing**

**1** Click the label to be moved and select the square grip.

**2** Move the label to the new location.

---

**NOTE**

To return the label to its original position, if the leader is not modified from its original state, click the
🔵 reset grip or right-click the label and click Reset Label. For group labels, click the 🔵 reset grip or right-click the label and click Reset All Group Labels.

If the leader has been divided into multiple segments, a single click will restore the leader to its original state. Click the 🔵 reset grip twice to reset the label to its original location.

---

## Editing Point Label Components with Grips

Move each component of a point label independently by using component grips.

The following illustration shows the round and diamond-shaped component grips on a point label:



**To edit point label text with grips**

1   Click the point label to display the round and square grips.

2   Click the round grip to display the diamond-shaped grips.

3   Click a diamond-shaped grip and move the label component to the new location.

## Editing Line, Arc, and Polyline Labels with Grips

Use the square grip on line, arc, and polyline labels to drag the label off the entity. Use the diamond grip to slide the label along the entity.

If you slide a label along a polyline from one type of segment to another (such as a line to a curve), the label style changes.

**To move line, arc, and polyline labels away from the entity**

1   Click the label to be moved and select the ▢ grip.
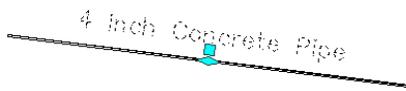


2   Move the label to the new location. A leader is created and the label is displayed using the Dragged State settings defined in the label style. A ⬭ grip is also displayed on the label.
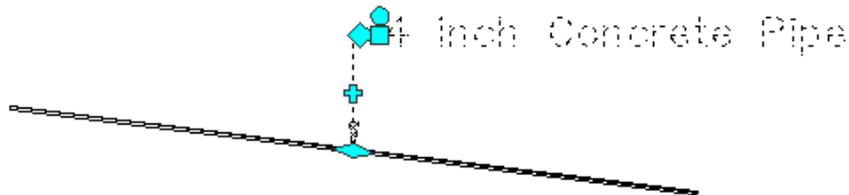


> **NOTE** If no additional leader line vertices have been added, click the ⬭ grip store a dragged label to its undragged state. If there are multiple leader lines, click the ⬭ grip once to restore the leader to its original state. Click the ⬭ grip twice to restore a dragged label to its undragged state.

**To move line, arc, and polyline labels along the entity**

1   Click the label to be moved and select the diamond grip.

2   Slide the label along the entity.

## Editing Unanchored Labels with Grips

Single point type labels such as note and station offset labels are not anchored to any specific object. You can move them freely about the drawing.

**To move a Note label**

1 Click the label to be moved and select the diamond grip.

2 Move the label to the new location.

**To drag a Note label away from its insertion point and create a leader**

1 Click the label to be moved and select the ■ grip.



2 Move the label to the new location. A leader is created and the label is displayed using the Dragged State settings defined in the label style. A ⬭ grip is also displayed on the label.
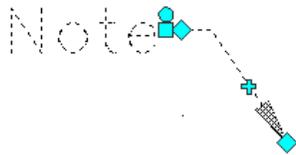


**NOTE** If no additional leader line vertices have been added, click the ⬭ grip to restore a dragged label to its undragged state. If there are multiple leader lines, click the ⬭ grip once to restore the leader to its original state. Click the ⬭ grip twice to restore a dragged label to its undragged state.

## Editing Data Band Labels with Grips

Edit data band label objects with grips.

Use the ■ grip on data band labels to drag the label off of the entity.

**To move a band label**

1 Click the label to be moved and select the ■ grip.

2 Move the label to the new location. A leader is created and the label is displayed using the Dragged State settings defined in the label style.

3 Manage the label leader as you would for other label objects. For more information, see Managing Leader Vertices (page 1520).

4 **NOTE** If no additional leader line vertices have been added, click the ⬭ grip to restore a dragged label to its undragged state. If there are multiple leader lines, click the ⬭ grip once to restore the leader to its original state. Click the ⬭ grip twice to restore a dragged label to its undragged state.

## Resetting Group Label Location

Returns dragged group labels to their original position.

Use the Reset All Group Labels command to return dragged group labels to their original position.

**To reset group label location**

1  Select one of the labels in the group.

2  Right-click the label, and click Reset All Group Labels.

> **NOTE** To reset a single sub-entity of a group label, the available command is Reset Label.

> **NOTE** You can also click the ⬤ grip to restore a dragged label to its default state. Click once to reset the leader to its original state if vertices have been added. Click twice to reset the label to its undragged state.

### Quick Reference

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Reset Label Group 🔲

Object Shortcut Menu

Right-click label ➤ Reset All Group Labels

## Resetting Single Label Location

Resets single labels to their original position.

Use this command to return dragged single labels to their original position. You can also reset a single, dragged sub-entity of a group label.
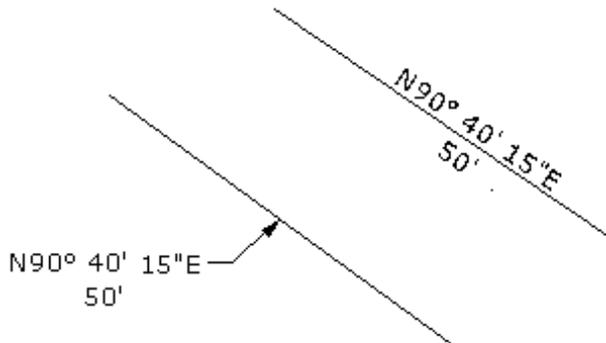


**To reset single label location**

1  Select the label you want to reset.

2  Right-click the label, and click Reset Label.

> **NOTE** You can also click the ⬤ grip to restore a dragged label to its default state. Click once to reset the leader to its original state if vertices have been added. Click twice to reset the label to its undragged state.

## Quick Reference

## Resetting Label Properties

Resets group labels.

Use this command to reset dragged group labels group back to their original position, as well as remove any style overrides.
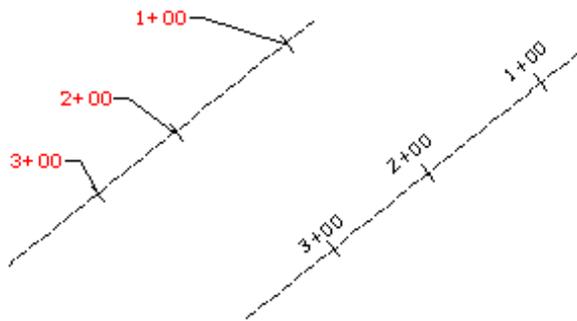


**NOTE** You can also reset dragged labels per viewport.

**To reset label group properties**

1 Select one of the labels in the group.

2 Right-click the label, and click Reset All Group Labels .

## Quick Reference

## Exploding Labels

Use the Explode command to convert labels to AutoCAD MText, line, and block entities without affecting the parent object.

If the label has a leader, then it is converted to an AutoCAD leader entity.

**To explode labels**

**1** Click Home tab ➤ Modify panel ➤ Explode  .

**2** Select the label to explode and press Enter.

## Quick Reference

Ribbon

Click Home tab ➤ Modify panel ➤ Explode 

Menu

Modify ➤ Explode

Command Line

Explode

# Flipping Labels

Flip a label in its location to reverse the placement order of the label components. You can flip labels for lines, curves, and spirals.

For example, flipping a bearing over distance label moves the bearing and distance to the other sides of the line or arc.
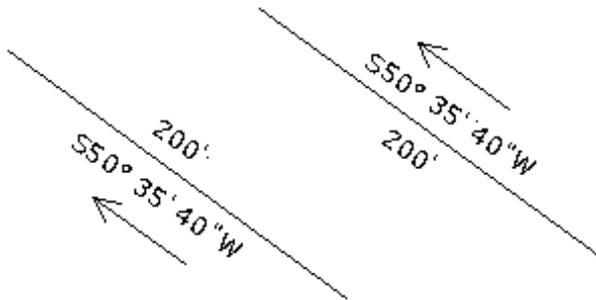


**To flip labels**

■ Right-click a label, and click Flip Label.

> **NOTE** To return the label to its original position, right-click the label and select Reset Label.

## Quick Reference

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Flip Label 

Object Shortcut Menu

Right-click label ➤ Flip Label

Command Line

  LabelFlip

## Pinning Labels

Pins labels to their current location.

After you have dragged a label to another location, you can force (pin) labels to remain in the current location, regardless of edits that are made to the parent object.

**To pin labels**

■  Right-click a label, and click Toggle Label Pin.

### Quick Reference

Ribbon

  Select the label. Click Labels tab ➤ Modify panel ➤ Toggle Label Pin

Object Shortcut Menu

  Right-click a label  ➤  Toggle Label Pin

Command Line

  LabelTogglePin

## Reversing Label Direction

Reverses label direction.

Reverse a bearing direction component in labels that support bearing direction.
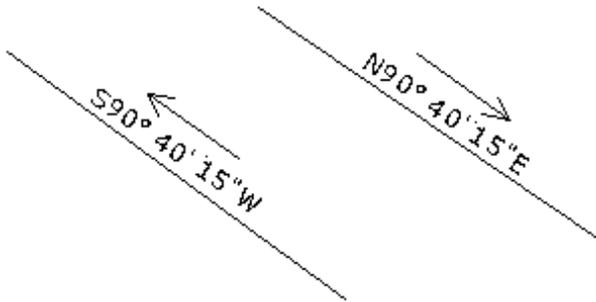


**To reverse a label direction**

■  Right-click a label, and click Reverse Label.

### Quick Reference

Ribbon

Select the label. Click Labels tab ➤ Modify panel ➤ Reverse Label

Object Shortcut Menu

Right-click label ➤ Reverse Label

Command Line

LabelReverseBearing

# Creating Label Tags

Control label tags with the Table Tag Numbering dialog box. You can set the default starting number and increment options for table tags.

General line and curve labels can be used alone or with parcel segment labels to create a table.

**To control table tag starting numbers and increments**

■ In the Toolspace Prospector tab, right-click the drawing name, and click Table Tag Numbering.

■ In the Table Tag Numbering (page 1971) dialog box, specify the default tag starting number and the increment number. No duplicate tags are created unless you manually create them with the Renumbering label tags tool. If there is a duplicate with the starting number, the number increments up by the specified amount. If there is still a duplicate, it increments up, until an unused tag number is found and applied. The starting number is then set to the next number according to the increment (even if it is a duplicate).

### Quick Reference

Object Shortcut Menu

Right-click drawing name ➤ Table Tag Numbering

# Renumbering Label Tags

Label tags are numbered sequentially according to the order in which they are created, but you can renumber them to remove gaps in a sequence, or to change the starting number or increment value.

By default, newly created tags are unique. Duplicate tags are only created manually. Specify tag starting and increment numbers in the Table Tag Numbering Dialog Box (page 1971). You can create a duplicate by opening an existing drawing that contains duplicates across sites/alignment collections or with the renumber command. You can create or avoid duplicate tags as desired.

Table tags support the following tag mode labels: general line, general curve, parcel line, parcel curve, alignment line, alignment curve, and alignment spiral.

**To renumber tags**

1 Click <Object Name> tab ➤ Labels & Tables panel ➤ Renumber Tags .

2 Do one of the following:

■ In the drawing, select a tag to renumber. The starting number is determined by the Table Tag Renumbering settings. If there is a duplicate in the drawing, you are prompted to skip to the next

available number or to create a duplicate. By default, duplicates are not created unless you allow it. You can select multiple tags to renumber. Select the tags in the sequential order that you want them numbered. If you selected a label that is not a valid type or is not in tag mode, you are prompted to try again.

■ At the command line, enter **S**. In the Table Tag Numbering (page 1971) dialog box, set the starting number and increment for the label types, either line, curve or spiral. Then press OK.

**3** Enter **end** to end the command.

### Quick Reference

Ribbon

Click <Object name> tab ➤ Labels & Tables panel ➤ Renumber Tags. 

Menu

<Object name> ➤ Add Tables ➤ Renumber Tags

Dialog Box

Renumbering (page 2444)

## Deleting Labels From Drawings

You can delete some labels from drawings.

Labels that you can delete include all label types that you can add manually, after an object is created. You can also delete group labels. Labels that you cannot delete include some of the label types that are added to an object automatically when it is created, such as parcel area labels.

**NOTE** You can also explode labels, and then erase the exploded entities. For more information, see Exploding Labels (page 1541).

If you delete an object, all labels that are associated with it are also deleted.

**To delete labels from drawings**

■ To delete manually-added labels, right-click a label and click Erase.

**NOTE** Some labels cannot be deleted, but can be hidden. For more information, see Hiding and Displaying Labels in the Drawing (page 1487).

### Quick Reference

Object Shortcut Menu

Right-click label ➤ Basic Modify Tools ➤ Erase

## Deleting Band Labels

You can delete band labels from drawings.

Band label objects can be deleted individually or as part of a group.

**To delete band labels as a group**

■  Right-click a band group label and click Erase.

**To delete individual band labels**

■  Ctrl-click a band label and click Erase.

### Quick Reference

Object Shortcut Menu

Right-click label ➤ Basic Modify Tools ➤ Erase

## Restoring Band Labels

You can restore band labels that have been deleted from drawings.

When the complete set of band group labels are deleted, the band displays without labels. You can use the Bands tab in the Profile View Properties dialog box to restore deleted band group labels.

The following workflows describe how to restore deleted profile view band labels, and the same concepts can be applied to restoring section view band labels.

**To restore deleted band group labels**

1  Right-click the profile view and select Profile View Properties.

2  In the Profile View Properties dialog box, click the Bands tab.

3  Select the Show Labels check box. Click OK.

**To restore deleted individual band labels**

■  Right-click a band label and click Reset All Group Labels
All previously deleted labels display on the bands.

## Deleting a Label Style

You can delete a label style if it is not being used in a drawing.

**To delete a label style**

1  In Toolspace, on the Settings tab, right-click the label style you want to delete.

2  Click Delete.

> **NOTE** You cannot delete a label style that is referenced by an object or by another label style.

3  Click Yes to confirm the deletion.

# Creating a Label Set

Defines new label sets.

This allows you to modify the set of labels by adding or removing label types or parameters.

**To define a new label set**

1   In Toolspace, on the Settings tab, expand a drawing's Alignment, Profiles, or Section object collection, and then expand Label Styles.

2   Right-click Label Sets, and click New.

3   In the <Feature Name> Label Set (page 1968) dialog box, click the Information tab.

4   Enter the label set name and description information.

5   Click the Labels tab.

6   Under Type, select a label style type from the list.

7   Under <label style name> Label Style, select a specific label style from the list.

8   Click Add to add the style to the label set.

    The label style is displayed in a row in the window section.

9   Repeat Steps 7–8 for each label style that you want to add to the set.

10  Specify values for the label properties, such as the Increment value for station labels. For more information, see Labels Tab (Label Set Dialog Box) (page 1968).

11  Click OK.

**Quick Reference**

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <Alignment, Profile, or Section object collection> ➤ Label Styles ➤ right-click Label Sets ➤ New

Dialog Box

<Feature Name> Label Set (page 1968)

# Expressions

Set up mathematical expressions to use in label styles.

Label styles are composed of property fields that display information about an object. Expressions allow you to convert the object information using a formula, then insert the result into a label style component.

Expressions are stored in the Settings tree, within the label style type folders. They are not available for label style types that do not have relevant properties, such as Note label styles.

You insert expressions into label styles using the Text Component Editor when you are editing a label style text component.

You can use design data to determine how a label style is drawn using text, lines or block components. You can insert expressions into label styles by selecting expressions in place of static values for text, line, and block component modifiers. For example, in the Layout tab of the Label Style Composer dialog box, you can insert an expression in place of a static value when determining the text height of a label. This expression could increase the text height of a spot elevation label to coincide with the elevation height.

You can insert expressions in label styles to create custom graphics. For example, you can create a graphical profile band, in which the graphics of the vertical grades are relative to the actual grades in order to visually reinforce the data representation, or match graphical standards. A PVI can be represented by a symbol, where the lines/values representing the grade in and grade out change quadrants based on the sign of the grades.

# Setting Up Expressions

Expressions make use of the same properties that you can add to label styles, such as Point Elevation, Northing, and Easting. By using expressions, you can set up separate mathematical formulas using the existing properties. For example, you could subtract a value from a point elevation, and display that number along side the actual elevation in a point label.

After you set up expressions, they are available in the Properties list in the Text Component Editor so you can add them to label styles. In effect, expressions become new properties that you can use to compose a label style.

Expressions are unique to a particular label style type. Only those properties that are relevant to the label style type are available to choose from in the Expressions dialog box.

### Brackets and Parentheses

When you insert a property into an expression, it appears in curly brackets if it contains spaces. For example:

```
{Segment Length}
```

Do not insert anything else into the curly brackets, and do not remove them. If the property name does not have any spaces (like Northing), then curly brackets are not required.

For expression grouping, use parentheses. For example:

```
0.567*({Segment Length} + 56)
```

### Restrictions

Expressions are mathematical formulas that are used to compute a value. No text can be added directly to them. Any text you want to include can be added to the label style. For example, to convert feet to meters, you could set up the following expression:

```
{Segment Length} *.3084
```

If you want to add "meters" to the end of this string, you must add it later when you set up the label style.

### To set up an expression

1   In Toolspace, on the Settings tab, expand a label style type collection.

    For example, expand Parcel ➤ Label Styles ➤ Line.

2   Right-click Expressions and click New.

3   In the Expressions dialog box, enter a name and description for the expression.

**NOTE** It is recommended that you use a clear naming convention such as "Parcel line expression: Length in meters" so you can distinguish them when composing a label style.

**4** Click the Insert Property button and select a property from the list.

The property appears in the editor box.

**5** Add functions, constants, and logical operators to complete the equation.

**NOTE** Do not insert anything into the curly brackets that surround the property fields.

■ Use the calculator buttons or click in the editor field and use your keypad to enter numbers and mathematical operators like **\*** (multiply) or \ (divide).

■ Use the Insert Function button to insert mathematical functions like TAN.

**NOTE** For a complete list of operators and functions, see Expressions Dialog Box (page 1973).

**6** Specify how the expression should be formatted by selecting a format from the Format Result As list.

**7** Click OK.

See Inserting Expressions into Label Styles as Properties (page 1551)

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object type collection> ➤ <label style type collection> ➤ right-click Expressions ➤ New

Dialog Box

Expressions (page 1973)

# Expression Examples

Following are some examples of expression syntax.

### Converting Northings and Eastings to Polar Coordinates

You can create expressions for point labels to convert northings and eastings to polar radius and polar angle values. These expressions compute the radius and angle from the coordinate system origin to the point.

**Polar Radius Expression Syntax:**

SQRT (Northing^2 + Easting ^2)

Format as Distance.

**Polar Angle Expression Syntax:**

ATAN2 (Northing, Easting)

Format as Angle.

### Converting Astronomic Direction to Magnetic Direction

All internal angle calculations in AutoCAD Civil are done in radians. You can take this into account when you apply a formula to an angle, such as when you label a line with both a true direction and a magnetic direction.

To subtract a 15d 30' 15" magnetic declination from an astronomic direction to derive the magnetic direction, you can set up the following expression:

**Known:**

PI = 3.141592654

360 degrees = 2(pi) radians

1 radian = 57.29577951 degrees

**Given:**

15d 30' 15" = 15.50417 degrees

**Expression Syntax:**

{Segment Direction} - (15.50417**\***((2**\***pi)/360))

Or alternatively using the built-in functions as follows:

{Segment Direction} - DEG2RAD(15.50417)

Format the result as Direction.

### Tower Top Elevation

You can create expressions for surface spot elevation labels to calculate the sum of the height of a structure such as an airport control tower with the ground elevation.

**Tower Top Expression Syntax:**

{Surface Elevation+ 228.5}

In this example, 228.5 is the height of the structure.

# Editing Expressions

After you create an expression, it is listed in the Settings tree item view.

Click the Expressions collection to display the list of defined expressions in the Toolspace item view. Right-click an expression to access commands.

**To edit an expression**

1  Select the Expressions collection that contains the expression you want to edit.

2  Ensure the Toolspace item view (page 82) is visible.

3  Right-click an expression in the item view and click Edit Expression.

4  Edit the expression using the Expressions (page 1973) dialog box.

# Inserting Expressions into Label Styles as Properties

Expressions are listed in the Properties list of the Text Component Editor. You can insert them into label styles like any other property.

**To insert an expression into a label style**

1  Set up an expression (page 1548) for a label style type.

2  Open an existing label style in the Label Style Composer or create a new label style. The type of this label style must match the type of the expression you created.

   For example, if you created an expression for a parcel line, you can only add that expression to a parcel line label style.

3  In the Label Style Composer dialog box, click the Layout tab and select or create a text, line, tick, or block component to which you want to add the expression. For more information, see Text (page 1502).

4  Under Text, click the Value column for Contents click [...]
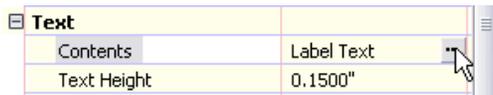


5  In the Text Component Editor, on the Properties (page 1962) tab, click the Down arrow on the Properties list.

   This list contains a set of properties and expressions applicable to each label style type.

6  Select the expression and click ➪. This inserts the expression into the Text Component Editor window.

7  Add any descriptive label text before or after the expression as needed. For example, add descriptive information about the expression, such as units, that will distinguish it in the label.

   **WARNING** Expressions have the same editing restrictions as property fields. If you alter anything about the expression, such as using the shortcut menu to change capitalization, the expression will not function correctly.

8  Click OK to close the Text Component Editor and the Label Style Composer dialog boxes.

### Quick Reference

Dialog Box

# Inserting Expressions into Label Styles as Values

You can insert expressions as values to modify components in the Label Style Composer.

**To insert an expression as a value into a label style**

1  In Toolspace, on the Settings tab, expand a label style collection. For example, expand Parcel ➤ Label Styles ➤ Line.

2  Right-click Line and click Edit.

3  In the Label Style Composer dialog box, click the Layout tab and click a component modifier to access a drop-down list of compatible expressions. You can also type in a new value.

4  Click OK.

> **NOTE** When inserting expressions as values for text, line, block or tick components, any non-number, or "double" type property cannot be used and will not appear in the drop-down list. Height, length, or size modifiers must be absolute numbers, so that the property is converted to an absolute value. For example, if the entered value for text height is zero, the default height from drawing label style defaults will be used.

The tables below list the Label Style Composer component modifiers that support expressions.

**Text Components**

Text Height

Rotation Angle

X Offset

Y Offset

**Line Components**

Length

Angle

Start Point X Offset

Start Point Y Offset

End Point X Offset

End Point Y Offset

**Block Component**

Block Height

Rotation Angle

| Block Component |
| --- |
| X Offset |
| Y Offset |

| Tick Components |
| --- |
| Block Height |
| Rotation Angle |

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Label Styles ➤ <label style type name> ➤ right-click <label style name> ➤ Edit

Dialog Box

Label Style Composer (page 1942)
Text Component Editor (page 1961)

# Labels Command Reference

The following table lists label commands and briefly describes their functionality. For more information about a command, follow the link in the Description column.

| Command | Description |
| --- | --- |
| AddLabels | Adds labels with the Add Labels dialog box (page 1522) |
| AddNoteLabel | Adds a note label (page 1530) |
| AddSegmentLabel | Adds line, arc, and polyline segment labels (page 1523) |
| FlipLabel | Flips a label to reverse the placement order of th e label components. (page 1542) |
| LabelReverseBearing | Reverses bearing direction in labels that support bearing direction. (page 1543) |
| LabelTogglePin | Pins labels to their current location. (page 1543) |

# Tables                                                          34

You can insert tables into a drawing to document the geometry of selected objects.

## Understanding Tables

Each row in a table contains information about a single object component, such as a line. Tables are supported for points, parcels, alignments, surfaces, pipes, and structures.

■ The first column of data in a table is reserved for the tag number or identifier.

> **NOTE** For point tables, the point number is used as the identifier. For parcel tables, the parcel ID is used as the identifier for the area and tag numbers are used for lines and curves.

■ Additional columns display the object data.

**Adding Object Tables**

Add a table to a drawing by doing the following:

■ Set up a table style for the object. The table style specifies the type of data that is inserted into the table.

■ Label the objects whose data you want to add to a table.

Select table data by using the following methods:

■ By selecting individual labels

■ By specifying a label style

■ By selecting point groups (for point tables)

■ By selecting an alignment (for alignment segment tables)

When you insert a table into a drawing, most labels are required to be displayed in tag mode (the exceptions to this rule are parcel area tables and point tables).

If the labels are currently set to label mode, they are automatically switched to tag mode when you insert the table. Although it is not required, you may prefer to set the label styles to tag mode before inserting a table to avoid the creation of unwanted styles. For more information, see Setting Up Label Styles To Be Used as Tags (page 1485).

After creating tags, you can renumber them to reset their sequential order. For more information, see Renumbering Label Tags (page 1544).

**Adding Surface Legend Tables**

When you perform a surface analysis, the data is separated into ranges that are assigned different colors. Legend tables display these colors and range values.

Add a surface legend table to a drawing, by doing the following:

- Perform a surface analysis. For more information, see Analysis Tab (Surface Properties Dialog Box) (page 2349).

- Modify the surface style so that it displays the analysis type. For more information, see Display Tab (Surface Style Dialog Box) (page 2345).

- Set up a surface legend table style to display the surface data. For more information, see Table Styles (page 1556).

# Table Styles

Edits table content and style.

Table styles define which data is displayed in the table, and control the table appearance.

# Defining a New Table Style

Right-click a table style type and click New to create a new table style.

**To define a new table style**

1 In Toolspace, on the Settings tab, right-click a table style type name, and click New.

2 In the Table Style dialog box, click the Information tab (page 2437).

3 Enter a name.

4 Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ Right-click <table style type name> ➤ New

Dialog Box

Table Style (page 2437)

# Table Text Properties

Define text properties for a table style, such as text style and height.

**To define text style**

1 In the Table Style dialog box, click the Data Properties (page 2437) tab.

**2** Under Text Settings, select text styles from the Title Style, Header Style, and Data Style lists.

**3** Under Text Settings, set text heights for the Title Style, Header Style, and Data Style text styles.

**4** Click OK.

### Quick Reference

Ribbon

Click Table tab ➤ Modify panel ➤ Table Properties drop-down ➤ Edit Table Style ▦

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Table Title

The title is displayed in the first row of a table.

---
**NOTE** Except for Alignment tables, you cannot define property fields as a table title.

---

**To define the table title**

**1** In the Table Style dialog box, click the Data Properties (page 2437) tab.

**2** Under Structure, double-click the table title element in the top row of the table heading. The Text Component Editor opens.

**3** Enter the title.

**4** Format the text using the editing options.

**5** Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Table Column Names

Column titles are displayed in the second row of a table.

**NOTE** The first column must be the tag column.

**To define column titles**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Under Structure, double-click a column heading to open the Text Component Editor.

3   Enter the column title.

4   Format the text using the editing options.

5   Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Table Data

Properties control which data is inserted in table columns.

Available properties are specific to the table type. For example, point table properties include only those properties that are relevant to points, such as point number.

Surface legend table styles contain a Surface Range Color property that displays the color swatch in the specified column.

**To link property fields to table rows**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Under Structure, double-click a Column Value cell to open the Text Component Editor.

3   Click the Properties tab.

4   In the Properties list, select a property, and enter the Modifier values.

5   Click  to insert the property into the window.

6   Click OK.

### Quick Reference

# Table Format

Manage table size and appearance.

## Column Size

Set columns to either a specific or an automatic width.

■ You can set the column width to Manual and specify the number of characters allowed in a string. Strings over the specified amount are truncated unless word wrapping is selected.

■ You can set the column width to Automatic, which adjusts the width to accommodate the longest string in the column.

**To define column size**

1 In the Table Style dialog box, click the Data Properties (page 2437) tab.

2 Under Structure, click a Column Width icon ⬚ to display the selections.

3 Click either Automatic or Manual.

4 If you select Manual, enter the number of characters.

5 Click OK.

### Quick Reference

## Rearranging Columns

Rearrange and move columns in the table by dragging them.

**To move columns**

**1** In the Table Style dialog box, click the Data Properties (page 2437) tab.

**2** Under Structure, click the column to be moved in its title cell and drag it to the new position.

---

**NOTE** You cannot move the first table column.

---

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

## Adding or Deleting Columns

Add or delete columns in a table.

**To add columns**

**1** In the Table Style dialog box, click the Data Properties (page 2437) tab.

**2** Under Structure, click ✛ to add a column.

**3** Double-click the blank column heading to open the Text Component Editor window.

**4** Enter a title, and then click OK.

**5** Double-click the blank Column Value cell and select a property on the Properties tab of the Text Component Editor. For more information, see Table Data (page 1558).

**To delete columns**

**1** In the Table Style dialog box, click the Data Properties (page 2437) tab.

**2** Under Structure, click the column and then click ✖ .

**3** Click OK.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

## Sorting Columns

Sort columns in ascending or descending order.

**To sort a column**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Select the Sort Data check box.

3   Select the column whose rows you want to sort and specify whether the sort should be in Ascending or Descending order.

4   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

## Splitting Tables

Split tables into smaller sets and customize the display of the headers.

**To set up a split table**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Under Table Settings, you can select the following options:
   - Repeat Title In Split Tables.
   - Repeat Column Headers In Split Tables.

3   Click OK.

**NOTE** When you create a table, or edit table properties, you can specify both whether the table should be split and how many rows each section should contain. For more information, see Table Properties Dialog Box (page 2443).

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

## Word Wrapping

If any columns are set to a Manual width, you can enable the Wrap Text option. This option wraps text that exceeds the character limit instead of truncating it.

**To enable word wrapping**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Under Table Settings, select the Wrap Text check box.

3   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

## View Orientation

Use the Maintain View Orientation option to ensure that tables appear plan-readable.

**To maintain the view orientation**

1   In the Table Style dialog box, click the Data Properties (page 2437) tab.

2   Under Table Settings, select the Maintain View Orientation check box.

3   Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Table Display

Specify layer, color, and linetype options for a table style.

### To define table display

1  In the Table Style dialog box, click the Display (page 2440) tab.

2  In the View Direction list, select either 2D or 3D.

3  Specify the display options for each component type.

4  Click OK.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Viewing a Summary of Table Settings

Review the settings for a table style.

### To view a summary of table settings

■  In the Table Style dialog box, click the Summary (page 2441) tab.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Edit

Object Shortcut Menu

Right-click table ➤ Edit Table Style

Dialog Box

Table Style (page 2437)

# Using Table Styles from Other Drawings

Drag and drop table styles from other drawings to copy the styles.

Drag a style from one drawing to another in the Settings tree, or drag the style from the Settings tree into the drawing window to copy the style into that drawing.

When you attempt to copy a table style to a location where a matching table style exists, you are prompted with a warning, and have the following options:

■ **No**. Leaves the existing style and cancels the operation.

■ **Yes**. Copies the style to the new drawing and renames the style using the format <source drawing name>.<style name>.1

**To copy table styles from other drawings**

1 Open the drawing that contains the styles you want to copy (Drawing1, for example), and the drawing you want to copy styles to (Drawing2, for example).

2 In Toolspace, click the Settings tab.

3 In the View Control list, select Master View.

4 In the Settings tree, expand the Drawing1 collection and select the table style to copy.

5 Drag and drop the table style to Drawing2 in the Settings tree. You must drag the style over the drawing name in the Settings tree before releasing the mouse button.

■ A valid drag and drop operation will display a ⬚ cursor.

■ An invalid drag and drop operation will display a ⊘ cursor.

# Deleting a Table Style

Use the Settings tree in Toolspace to delete a table style.

**To delete a table style**

■ In Toolspace, on the Settings tab, expand the object collection, expand the table style type collection (if applicable), right-click the table style name, and click Delete.
For example, to delete the surface direction table style named 'Standard':

NOTE A style that is used in the drawing cannot be deleted.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: <drawing name> ➤ <object collection> ➤ Table Styles ➤ <table style type name> ➤ Right-click <table style name> ➤ Delete

# Adding Tables to Drawings

Add tables to a drawing.

# Adding Object Tables to Drawings

Specify table data differently for each object type.

- **Point tables**. Select points, point groups, or label styles.

- **Alignment tables**. Select an alignment, individual labels or tags, or label styles.

- **Parcel tables**. Select individual parcel labels or tags, or label styles.

- **Survey figure tables**. Select survey figure labels or tags, or label styles.

When creating parcel tables, you can select any general line/curve label types and/or parcel segment labels including any line or curve label, applied to a line, curve, feature line, or polyline.

You can create an alignment segment label that references a general line or curve style for inclusion in a table.

General line and curve label styles have a Tag Mode which supports general line and curve labels, parcel segment labels that reference a general line or curve style, and alignment segment labels that reference a general line or curve style.

The Add Table commands are available from the Annotate tab.

**To add tables to a drawing**

1   Do the following:
    ■   Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ <feature> ➤ Add <feature-specific> Table ⊞ .

2   In the Table Creation dialog box, select the table style, layer, and behavior properties. For more information, see Table Creation Dialog Box (page 2441).

3   Select the data for the table:
    ■   For all types of tables, select the check boxes next to the style name to select label styles to include in the table.

    ■   For alignment segment tables, select By Alignment and click ⬛ to select an alignment. Or, select Multiple Selection and select the check boxes or click ⬛ to select tags.

    ■   For alignment line, curve, or spiral tables, click ⬛ to select tags.

    ■   For point tables, click ⬕ to specify a point group or click ⬛ to select points from the drawing.

    ■   For parcel tables, click ⬛ to select tags or area labels. Note that you can select line and curve labels within the drawing, just as you would select parcel segment or alignment labels. These labels go into tag mode when selected. If they are selected in the drawing, you are prompted to create a child style in tag mode for the selected label.

    **NOTE** To avoid creating child label styles, set up label styles to be used as tags. For more information, see Setting Up Label Styles To Be Used as Tags (page 1485).

4   Click OK.

5   Select an insertion point.


## Quick Reference

Ribbon

Click Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ <feature> ➤ Add <feature-specific> Table ⊞

Menu

<Feature> menu ➤ Add Tables

Dialog Box

Table Creation (page 2441)


# Adding Surface Legend Tables to Drawings

Surface tables are created in a legend style and do not use label styles or tags.

To add a surface legend table to a drawing, first do the following:

■ Perform a surface analysis. For more information, see Analysis Tab (Surface Properties Dialog Box) (page 2349).

■ Modify the surface style so that it displays the analysis type. For more information, see Display Tab (Surface Style Dialog Box) (page 2345).

■ Set up a surface legend table style to display the surface data. For more information, see Table Styles (page 1556).

**To add surface legend tables**

1 Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Add Surface Legend Table ⊞

2 Select a surface in the drawing.

3 At the Command prompt, enter the table type.

4 At the Command prompt, enter Dynamic to enable the table to be updated automatically if the surface analysis is updated. Enter Static to require manual updating.

5 Select an insertion point.

### Quick Reference

Ribbon

Annotate tab ➤ Labels & Tables panel ➤ Add Tables menu ➤ Add Surface Legend Table ⊞

Menu

Surfaces menu ➤ Add Legend Table

Dialog Box

Table Creation (page 2441)

# Modifying Tables

You can modify the appearance of a table, or add or remove data.

# Changing Table Properties

Use to edit table properties.

Change table properties to modify table content, appearance, layout, and update behavior.

**To change the properties of a table in a drawing**

1 Right-click a table, and click Table Properties.

2 In the Table Properties (page 2443) dialog box, you can change the Table Style, Split Table, and Behavior options.

3 Click OK.

### Quick Reference

Ribbon

Click Table tab ➤ Modify panel ➤ Table Properties drop-down ➤ Table Properties 

Object Shortcut Menu

Right-click table ➤ Table Properties

Dialog Box

Table Properties (page 2443)

# Changing Table Data

Add, remove, or replace data in a table.

___

**NOTE** The Table Selection shortcut commands are not available for surface legend tables and alignment segment tables.

___

**To add data to a table**

1   Right-click a table, and click Selection ➤ Add.

2   In the Selection dialog box, select the data by doing the following:

  ■ **All table types**: Select label styles by selecting the Apply check box and then selecting Add Existing or Add Existing And New from the Selection Rule list.

  ■ **Point tables**: Click  and select point groups, or click  and select points in the drawing.

  ■ **Alignment tables**: Click  and select alignment tags in the drawing.

  ■ **Parcel tables**: Click  and select parcel tags in the drawing.

3   Click OK.

**To remove data from a table**

1   Right-click a table, and click Selection ➤ Remove.

2   In the Selection dialog box, select the data by doing the following:

  ■ **All table types**: Select label styles by selecting the Apply check box, or click  and select table rows.

  ■ **Point tables**: Click  and select point groups.

3   Click OK.

**To replace data in a table**

1   Right-click a table, and click Selection ➤ Replace.

**2** In the Selection dialog box, select the data by doing the following:

- **All table types**: Select label styles by selecting the Apply check box and then selecting Add Existing or Add Existing And New from the Exclusion Rule list.

- **Point tables**: Click ⊞ and select point groups, or click ⊞ and select points in the drawing.

- **Alignment tables**: Click ⊞ and select alignment tags in the drawing.

- **Parcel tables**: Click ⊞ and select parcel tags in the drawing.

**3** Click OK.

### Quick Reference

Ribbon

Select the table. Click Table tab ➤ Modify panel ➤ Add Items ⊞
Click Table tab ➤ Modify panel ➤ Remove Items ⊞
Click Table tab ➤ Modify panel ➤ Replace Items ⊞

Object Shortcut Menu

Right-click table ➤ Selection ➤ Add
Right-click table ➤ Selection ➤ Remove
Right-click table ➤ Selection ➤ Replace

Dialog Box

Add Selection (page 2444)
Remove Selection (page 2445)
Replace Selection (page 2445)

## Updating Table Data

Use to update table content.

Set to Static mode to keep data from updating dynamically.

If a table is set to Static behavior mode, you can update it in the Table Properties dialog box or use the Update Content command.

The Update Content command overrides static mode and updates the table data.

**To update table data**

**1** Right-click a table and click Table Properties.

**2** In the Table Properties dialog box, click Force Content Update.

> **NOTE** You can select this option only when the Reactivity Mode is set to Static.

**3** Click OK.

OR

➤ Right-click a table and click Update Content.

### Quick Reference

Ribbon

Select the table. Click Table tab ➤ Modify panel ➤ Update Content 

Object Shortcut Menu

Right-click table ➤ Table Properties
Right-click table ➤ Update Content

Dialog Box

Table Properties (page 2443)

# Locking Table Data

Use to set table content to static mode.

Set to Static mode to keep data from updating dynamically.

The Update Content command overrides static mode and updates the table data.

### To set table data to static mode

➤ Right-click a table and click Static Mode.

### Quick Reference

Ribbon

Select the table. Click Table tab ➤ Modify panel ➤ Static Mode 

Object Shortcut Menu

Right-click table ➤ Static Mode

# Moving Tables

Use the grip in a table's upper-left corner to move it.

### To change the location of a table

■ Select the table and use its grip to drag it to a new location.

# Realigning Table Stacks

Use to realign table stacks.

Realigning a table restores table stacks to their original positions. It also adjusts columns if the table data has changed and a column has overlapped another table section.

**To realign table stacks**

1 Click any section of a table to select it.

2 Right-click and click Realign Stacks.

## Quick Reference

Ribbon

Select a table. Click Table tab ➤ Modify panel ➤ Realign Stacks ▣

Object Shortcut Menu

Right-click table ➤ Realign Stacks

# Joining Split Tables

Re-connect tables that were split.

**To join split tables**

1 Right-click a table and click Table Properties.

2 In the Table Properties dialog box, clear the Split Table check box.

3 Click OK.

## Quick Reference

Object Shortcut Menu

Right-click table ➤ Table Properties

Dialog Box

Table Properties (page 2443)

# Exploding Tables

Use the AutoCAD EXPLODE command to explode a table into an AutoCAD unnamed block.

An exploded table retains its appearance. However, if you use EXPLODE again, then table components are converted to AutoCAD objects as follows:

| These exploded table components... | ...are converted to these AutoCAD objects. |
| --- | --- |
| text | MText |
| boundary | polyline |
| interior borders | lines |
| boundary fills | hatch objects |

**To explode a table**

**1** Click Home tab ➤ Modify panel ➤ Explode  .

**2** Select the table and press Enter.

## Quick Reference

Ribbon

Click Home tab ➤ Modify panel ➤ Explode  .

Menu

Modify ➤ Explode

Command Line

EXPLODE

# Deleting Tables

Delete a table in a drawing.

**To delete a table from a drawing**

■ Right-click a table and click Erase.

## Quick Reference

Object Shortcut Menu

Right-click table ➤ Erase

# Reports

# 35

In AutoCAD Civil 2010, all reports are consolidated within the Reports Manager. The Reports Manager is displayed on the Toolbox tab of Toolspace.

## Understanding the Reports Manager

Use the Reports Manager to access all the report types that are available in AutoCAD Civil. You can add custom reports to the Reports Manager so even your custom reporting can be done through the same interface.

Reports in AutoCAD Civil are of two types: LandXML and .NET. Depending on the type of report, different dialog boxes are displayed after you select the report to run.

The LandXML reports use LandXML data output for report generation, and use predefined or custom XSL style sheets to format the report data.

The .NET reports use custom dialog boxes for selecting the data and options for the report.

The LandXML Reporting tool is included with AutoCAD Civil as a program that you can run independently of AutoCAD Civil. To use this tool, you export a LandXML file and then open it within LandXML Reporting. Access LandXML Reporting from the desktop icon or by selecting it from the AutoCAD Civil 2010 Program Group from the Start ➤ Programs menu. For more information about exporting LandXML data, see LandXML Import and Export (page 1673).

## Specifying Report Settings

Use the Edit Report Settings dialog box to specify report settings. These settings are used for all reports.

**To specify report settings**

1 Click View tab ➤ Palettes panel ➤ 🧰 Toolbox.

2 On the Toolbox tab, click 🧰 .

3 In the Edit Report Settings (page 2212) dialog box, expand the tree view and edit the values in the Value column.

4 Click OK.

**Quick Reference**

Ribbon

Click View tab ➤ Palettes panel ➤  Toolbox

Menu

General menu ➤ Reports Manager

Command Line

ReportsMgr

Dialog Box

Edit Report Settings (page 2212)

# Generating a Report

Generate reports by double-clicking a report in the Reports Manager.

**To generate a report**

1 Click View tab ➤ Palettes panel ➤  Toolbox.

2 Expand Reports Manager ➤ <Report Type> to see the available reports.

3 Double-click a report, or right-click a report and click Execute.

4 Do one of the following:

■ **LandXML Reports**: In the Export To LandXML (page 1983) dialog box, select the data to include in the report and click OK.

> **NOTE** If you are running a points report, for example, only the points that are selected are used in the report. You can clear the selection of other objects, but that is not necessary. All objects other than points are filtered out.

"LandXML export complete" and "Generating XSLT Report..." are displayed at the command line. The report is then displayed in the default Internet browser window.

> **NOTE** LandXML reports are not saved automatically. To save a LandXML report, see Saving Report Files (page 1575).

■ **.NET Reports**: Select the data to include and specify the location to save the report. For more information on the specific dialog boxes, see Reports Dialog Boxes (page 2211). Click Create Report. The report is then displayed in the default Internet browser window and saved to the specified location.

**Quick Reference**

Ribbon

Click View tab ➤ Palettes panel ➤  Toolbox

Menu

General menu ➤ Reports Manager

Command Line

ReportsMgr

# Saving Report Files

You can save your report for future use.

**NOTE** .NET reports are saved automatically to the specified location in the report dialog box.

**To save report files**

1  In the Internet browser window, click File menu ➤ Save As.

2  Change the folder to an appropriate location, or use the default, which is the last folder used.

3  Enter the file name and specify the file type for the report.

4  Click Save.

**NOTE** If you do not want to save the entire report, select the area of the report you want to save, copy it and paste the data into Microsoft Word, Microsoft Excel, or another editor. If you save your files as comma or space delimited files you can successfully import them into Microsoft Excel spreadsheets.

## Quick Reference

Menu

File menu ➤ Save As

Command Line

SAVEAS

# Editing Toolbox Content

The Reports Manager is displayed on the Toolbox tab of Toolspace. You can add custom reports to the Reports Manager, or other custom tools to the Toolbox.

**To edit Toolbox content**

1  Click View tab ➤ Palettes panel ➤ 🧰 Toolbox.

2  On the Toolbox tab, click 📝 .

3  To edit the Reports Manager items, in the Toolbox Editor vista (page 2211), expand the Reports Manager item.

4  Do one of the following:

■  To add a new collection for reports, right-click the Reports Manager item and click New Category. You can specify a name and description for the category by selecting the new category and editing the values displayed in the right pane.

■ To add a new report, right-click a collection and click New Tool.
A new tool is inserted into the category with a default name. Expand the collection and click the New Tool item to select it. Specify the properties for the new tool by editing the values in the right pane.

**Quick Reference**

Ribbon

Home tab ➤ Palettes panel ➤ ⏷ ➤ Toolbox 🧰

Menu

General menu ➤ Toolbox

Command Line

TOOLBOX

# Reports Command Reference

Quickly access reports functionality from the command line.

| Command | Description |
| --- | --- |
| ReportsMgr | Opens and closes the Reports Manager (page 1574) |
| Toolbox | Opens and closes the Toolbox tab of Toolspace (page 101) |

# Object Rendering

# 36

You can create realistic rendering of AutoCAD Civil objects using render materials.

A render material corresponds to a specific material in a civil model, such as asphalt, concrete, or gravel. It consists of a pointer to a material and settings that control the material properties when it is rendered.

You can apply render materials to several objects, including surfaces and surface masks, corridor surfaces and corridor surface boundaries, pipes and structures, and interferences.

For more information about render materials, including how to create custom render materials, modify render materials, add render materials to the drawing, and apply render materials to objects, see *Materials and textures* in the AutoCAD User's Guide.

## Creating Render Materials

Create render materials for enhancing the rendered views of surfaces, corridor models, pipes, structures, and interferences.

You use a render material to add a realistic look to an object in AutoCAD Civil. The material you select for visualization must exist in the current drawing.

After you create a render material, you can save it to a drawing template or copy it to other drawings.

If, at rendering time, the material specified as the render material cannot be found in the drawing, the AutoCAD GLOBAL material is used.

**To create a render material**

- To open the Materials window, click View menu ➤ Render ➤ Materials.

For instructions about how to create a render material, see *Create Materials* in the AutoCAD User's Guide.

### Quick Reference

Menu

  View menu ➤ Render ➤ Materials

Command Line

  MATERIALS

# Adding Materials to Drawings

To use a material in a rendering, the material must exist in the current drawing.

A Material Library is installed with AutoCAD Civil by default. You can open the Tools Palette and either drag a render material into a drawing or insert the material by using the Add to Current Drawing command.

**To add materials to the current drawing**

1   Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

2   In the Tool Palettes window, right-click the Tool Palettes Window button, and choose Civil Materials from the list.

---

**NOTE** In addition to civil-specific render materials, you can access and use all the render materials contained in the AutoCAD Materials Library palette. Choose one of the Materials Library palettes from the Tools Palettes Window shortcut menu.

---

3   In the Civil Materials palette, do one of the following:
   ■   Right-click the swatch representing the chosen render material and select Add to Current Drawing.

   ■   Drag and drop a render material swatch into the drawing.

   The selected render material is added to the drawing. For more information, see "Materials Dashboard and Tools Palettes" in AutoCAD Help.

## Quick Reference

Menu

General ➤ Tool Palettes Window ➤ click the Properties button ➤ choose Materials Library ➤ right-click a render material swatch ➤ Add to Current Drawing

Command Line

ToolPalettes

# Applying Render Materials to Objects

You can apply a render material to AutoCAD Civil objects.

You can specify a default render material to use for surfaces in the Surface Feature Settings (page 2329). After you create a surface or a surface mask, you can assign a different render material by changing the surface properties or surface mask properties.

Define the render material for corridor surfaces on the Surfaces tab (page 1818) of the Corridor Properties dialog box.

Specify the style for corridor boundaries on the Boundaries tab (page 1820).

Specify the styles for pipe network parts by using the Render Material option on the Pipe Properties Information tab (page 2039) or the Structure Properties Information tab (page 2052). Specify the styles for interferences by using the Interference Properties Information tab. (page 2073).

---

**TIP** Use corridor boundaries to apply different render materials to one corridor surface. For more information, see Rendering Corridor Models (page 1374).

---

**To apply a render material to an object**

**1** Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

**2** In the Tool Palettes window, click the Properties button, and choose Civil Materials from the list.

---

**NOTE** In addition to civil-specific render materials, you can access and use all the render materials contained in the AutoCAD Materials Library palette. Choose Materials Library from the Propertied menu.

---

**3** Do one of the following:

■ Right-click on the swatch representing the chosen material and select Apply Material to Objects.

■ Drag and drop a render material swatch onto the object in the drawing.

The selected render material is added to the drawing. For more information, see *Materials Tool Palettes* in the AutoCAD Help.

## Quick Reference

Ribbon

Home tab ➤ Palettes panel ➤ Tool Palettes Window

Menu

General menu ➤ Tool Palettes Window ➤ click the Properties button ➤ choose Materials Library ➤ right-click a render material swatch ➤ Apply Material to Objects

Command Line

ToolPalettes

# Rendering Objects

You can use the Render command to render objects.

**To render objects**

**1** Open the Materials window using the MATERIALS command.

**2** In the Materials window, apply a render material to the object you want to render. For more information, see Applying Render Materials to Objects (page 1578).

**3** Set the rendering properties. See *Basics of Rendering* in AutoCAD help.

**4** If you are rendering a corridor, create one or more corridor surfaces and corridor surface boundaries, and optionally export the corridor surface. For more information, see Exporting Corridor Surfaces (page 1373).

**5** To render the object in AutoCAD, click View menu ➤ Render ➤ Render.

## Quick Reference

Menu

View menu ➤ Render ➤ Render

Command Line

MATERIALS
RENDER

# Transparent Commands

# 37

Use AutoCAD Civil transparent commands to enter a value based on known information when you are prompted for a point, a distance, or a radius.

Most AutoCAD Civil transparent commands are used to specify point locations within a larger operation, such as the creation of an alignment or a parcel lot line. Using these transparent commands, you can calculate the location for a point from information, such as angle and distance, or from point object information, such as a point number.

**NOTE** For more information about AutoCAD transparent commands see AutoCAD Help.

Some of the AutoCAD Civil transparent commands require a Z coordinate for the point elevation. For more information, see Transparent Command Settings (page 1581).

## Transparent Command Settings

Use Transparent Command settings to control how you are prompted when you use a Civil transparent command.

The Transparent Command settings are specified on the Ambient Settings tab in the Edit Drawing Settings dialog box. In addition to the specific Transparent Command settings, many of the transparent commands are affected by other settings on the Ambient Settings tab.

**NOTE** Angle, Direction, and Lat Lon ambient settings are set to a single unit type (degree, grad or radian) and format (digital or various degree/minute/second formats for degrees). You will have to change these settings, if you wish to use a different unit/format for angles, bearings or latitudes/longitudes.

You can use the Transparent Command settings to control the following prompts for transparent commands:

■ Whether you are prompted for 3D points

■ The prompt order for X/Y

■ The prompt order for northing/easting

■ The prompt order for latitude/longitude

The Transparent Command settings are drawing ambient settings that can be overridden at either the object (feature) level or at the individual command level. For more information, see Ambient settings (page 68).

# Activating a Transparent Command

You can activate transparent commands by entering a command or by using a toolbar.

■ Enter the name of the transparent command at the command line. A transparent command always begins with an apostrophe (').

■ Select the command from the toolbar. To display the toolbar for accessing transparent commands, right-click any toolbar and select Transparent Commands.

# Using Transparent Commands Within a Running Command

You can use transparent commands within a main command that requires the entry of multiple point locations.

When you enter a transparent command within a running command that accepts more than one point location (for example, the AutoCAD LINE command), you can enter a series of points in the same format without having to re-enter the transparent command. At any time, you can press Esc to terminate the transparent command and return control to the main command, where you can switch to another transparent command. Also, you can end the transparent command by terminating the main command.

### Using Previously Entered Points in a Running Command for Relative Data Input

When you use a transparent command that requires baseline data from which to calculate a relative location, you must enter the required data before specifying the command parameters. If you use a transparent command within a running command, the transparent command remembers the last points entered during the running command and uses them to calculate new relative locations.

For example, when you use the Bearing Distance ('BD) transparent command, you must first identify a baseline point from which the bearing and distance are measured. If you initiate the Bearing Distance command before you enter any points, you are prompted for the baseline point before you are prompted for quadrant, bearing, and distance. If you use the Bearing Distance command after you have entered at least one point, the last point you entered is used as the baseline.

### Transparent Command Point Filters

When you are working within a transparent command and you are prompted to specify a point location, you can use point filters to specify known point information by doing one of the following:

■ Entering a point number

■ Entering northing and easting

■ Selecting a point in a drawing

For more information, see Using Point Filters Within Transparent Commands (page 1599).

# Using Angle and Distance to Specify a Point Location

You can specify a point location using an angle and a distance from a reference line.

The last two entered points define the reference line, or you must define a line. From the reference line (1-2), specify an angle (3), and distance (4), to specify a point (5).

If you have already entered one point in the command, you are prompted to enter a second point to define the reference line.

A transparent command can only be used within a running command. The reference line is updated each time you enter a point.

**To specify a point location using angle and distance**

1   When you are prompted to enter a point within a command, do one of the following:
    ■   Enter '**ad**.

    ■   Activate the Transparent Commands toolbar and select  .

2   If you have not yet entered any points in the main command, you must specify a temporary reference line by doing one of the following:
    ■   Select a line.

    ■   Enter **p**, then specify a start point and endpoint for the line.

3   If you have entered one point in the main command, that point is used as the starting point for the temporary reference line. Specify the ending point for the temporary reference line.

4   Specify the angle by doing one of the following:
    ■   Click in the drawing to specify the angle.

    ■   Enter a positive or negative numeric value for the angle.

    ■   Enter **c** to switch the angle between counterclockwise and clockwise, and then specify the angle.

5   Specify the distance by either clicking in the drawing or entering a distance.

6   If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

## Quick Reference

Command Line

'AD

Toolbar

Transparent Commands toolbar ➤ 

# Using Bearing and Distance to Specify a Point Location

You can specify a point location using a bearing and a distance from a reference point.

The last entered point is used, or you must specify a current point. From the start point (1), specify the quadrant (2), a bearing (3), and distance (4), to specify a point (5).



A transparent command can only be used within a running command. The reference point is updated each time you enter a point.

**To specify a point location using bearing and distance**

1   When you are prompted to enter a point within a command, do one of the following:
    ■   Enter '**bd**.

    ■   Activate the Transparent Commands toolbar and select  .

2   If you have not yet entered any points in the main command, specify a temporary point from which the bearing and distance will be measured.

3   Specify a quadrant number by either clicking in the drawing or entering a value between 1 and 4.

4   Specify the bearing within the quadrant by clicking in the drawing or entering a bearing using the angular units for the drawing.

5   Specify the distance by either clicking in the drawing or entering a distance.

6   If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

## Quick Reference

Command Line

'BD

Toolbar

Transparent Commands toolbar ➤ ◹

# Using Deflection and Distance to Specify a Point Location

You can specify a point location using a deflection angle and a distance from a reference line.

The deflection angle is measured from the reference line's extension beyond the endpoint, and the distance is measured from the line's endpoint

The extension of the last two entered points defines the reference line, or you must define a line. From the reference line (1-2) and start point (2), specify a deflection angle (3), and distance (4) to specify a point (5).



A transparent command can only be used within a running command. The reference line is updated each time you enter a point.

**To specify a point location using deflection and distance**

1 When you are prompted to enter a point within a command, do one of the following:
   ■ Enter '**dd**.

   ■ Activate the Transparent Commands toolbar and select ◹ .

2 If you have not yet entered any points in the main command, you must specify a temporary reference line by doing one of the following:
   ■ Select a line.

   ■ Enter **p**, then specify a start point and endpoint for the line.

3 If you have entered one point in the main command, that point is used as the starting point for the temporary reference line. Specify the ending point for the temporary reference line.

4 Specify the angle by doing one of the following:
   ■ Click in the drawing.

   ■ Enter a positive or negative numeric value for the angle.

   ■ Enter **c** to switch the angle between counterclockwise and clockwise, and then specify the angle.

5 Specify the distance by either clicking in the drawing or entering a distance.

6 If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

## Quick Reference

Command Line

'DD

Toolbar

Transparent Commands toolbar ➤ 

# Using Azimuth and Distance to Specify a Point Location

You can specify a point location using an azimuth and a distance from a reference point.

The last entered point is used, or you must specify a current point. From the start point (1) and North meridian (2), specify an azimuth (3), and distance (4), to specify a point (5).



A transparent command can only be used within a running command. The reference point is updated each time you enter a point.

**To specify a point location using azimuth and distance**

1   When you are prompted to enter a point within a command, do one of the following:
    ■   Enter '**zd**.

    ■   Activate the Transparent Commands toolbar and select  .

2   If you have not yet entered any points in the main command, specify a temporary point from which the azimuth and distance will be measured.

3   Specify an azimuth by either clicking in the drawing or entering a value.

4   Specify the distance by either clicking in the drawing or entering a distance.

5   If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

Command Line

ʻZD

Toolbar

Transparent Commands toolbar ➤

# Using Side Shots to Specify a Point Location

You can specify a point location using an angle (bearing, azimuth, angle, or deflection) and a distance from a reference point.

Before you can specify an angle and distance within the Side Shot command, you must define a temporary reference line from which all the subsequent points are calculated.

The last two entered points define the reference line, which is reused if you take multiple shots within the command. Specify an angle and distance to the next point. From the reference line (1-2) and start point (2), specify points (3, 4, 5) by side shot.



To define a line, you can either select an existing line in the drawing or enter two points. If you have already entered one point in the command, you are prompted to enter a second point to define the reference line.

A transparent command can only be used within a running command. In this command, the reference line is not updated; it remains the same for all points entered.

**To specify side shot point locations**

1 When you are prompted to enter a point in a command, do one of the following:

■ Enter ʼ**ss**.

■ Activate the Transparent Commands toolbar and select .

2 If you have not yet entered any points in the main command, you must specify a temporary reference line by doing one of the following:

■ Select a line.

■ Enter **p**, then specify a start point and endpoint for the line.

3 If you have entered one point in the main command, that point is used as the starting point for the temporary reference line. Specify the ending point for the temporary reference line.

**4** Enter an angle by doing one of the following:

- Enter an angle in angular units for the drawing.

- Enter **b**, then specify the angle by specifying a quadrant and a bearing.

- Enter **d**, then specify a deflection angle.

- Enter **z**, then specify an azimuth.

- Enter **c** to switch the direction of the angle between clockwise and counterclockwise.

**5** Specify a distance by either clicking in the drawing or entering a distance.

**6** If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

**7** If you are working within a running command, do one of the following:

- To enter another side shot using the same angle type, enter the angle and repeat Steps 5 and 6.

- Press Esc to end the Side Shot transparent command.

Additional side shot locations are calculated using the reference line established at the start of the transparent command.

### Quick Reference

Command Line

'SS

Toolbar

Transparent Commands toolbar ➤ 

# Using Northing and Easting to Specify a Point Location

You can specify a point location using known northing and easting values.

A transparent command can only be used within a running command.

**To specify a point location by northing and easting**

**1** When you are prompted to enter a point in a command, do one of the following:

- Enter '**ne**.

- Activate the Transparent Commands toolbar and select .

**2** Enter a northing value.

**3** Enter an easting value.

---

**NOTE** The order in which you are prompted for northing and easting is controlled by the Transparent Command setting Prompt For Easting Then Northing. For more information, see Transparent Command Settings (page 1581).

---

**4** If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

> **NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

### Quick Reference

Command Line

'NE

Toolbar

Transparent Commands toolbar ➤ 

# Using Grid Northing and Easting to Specify a Point Location

You can specify a point location using grid northing and easting values.

Grid settings are required in the drawing, including the zone, the coordinate system, and any applied transformations.

Transformation settings for the drawing may include a base point, a rotation angle, or another reference point.

A transparent command can only be used within a running command.

**To specify a point location by grid northing and grid easting**

**1** When you are prompted to enter a point in a command, do one of the following:

- Enter '**gn**.

- Activate the Transparent Commands toolbar and select  .

**2** Enter a grid northing value.

**3** Enter a grid easting value.

**4** If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

> **NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

### Quick Reference

Command Line

'GN

Toolbar

Transparent Commands toolbar ➤

# Using Latitude and Longitude to Specify a Point Location

You can specify a point location, using latitude and longitude values.

Zone and coordinate system settings are required in the drawing, and any applied transformations.

A transparent command can only be used within a running command.

**To specify a point location by latitude and longitude**

**1** When you are prompted to enter a point, do one of the following:

■ Enter '**ll** (two lower case Ls).

■ Activate the Transparent Commands toolbar and select ⊕ .

**2** Enter a latitude value.

**3** Enter a longitude value.

> **NOTE** The order in which you are prompted for latitude and longitude is controlled by the Transparent Command setting Prompt For Longitude Then Latitude. For more information, see Transparent Command Settings (page 1581).

**4** If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

> **NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

## Quick Reference

Command Line

'LL

Toolbar

Transparent Commands toolbar ➤ ⊕

# Using a Point Number to Specify a Point Location

You can specify a point location using a point number.

The system searches first in the active drawing, and then in the project, if applicable. You can enter multiple point numbers or number ranges separated by commas.

Only the northing and easting values from the specified point are used. If you want elevation values, you must enter or select an elevation.

You can use the command to specify a single point number, or you can specify multiple points by entering individual point numbers or ranges of point numbers separated by commas. For example, you could enter the following: 100-105, 603, 701-713.

A transparent command can only be used within a running command.

**To specify a point location by point number**

**1** When you are prompted to enter a point in a command, do one of the following:

- Enter '**pn**.

- Activate the Transparent Commands toolbar and select .

**2** Enter a point number or a list of point numbers.

**3** If you are prompted, enter an elevation.

If the point you specified in the previous step has an elevation, it is listed as the default elevation, and you can select it by pressing Enter. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

### Quick Reference

Command Line

'PN

Toolbar

Transparent Commands toolbar ➤ 

# Using a Point Object to Specify a Point Location

You can specify a point location by selecting an existing point.

Points are highlighted as you move the cursor over them.

The northing and easting value from the point are read from the selected drawing point. If you are prompted to enter an elevation, the default choice is the elevation value of the point. You can use the point's elevation value or you can enter another value.

A transparent command can only be used within a running command.

**To specify a point location by selecting a point**

**1** When you are prompted to enter a point in a command, do one of the following:

- Enter '**po**.

- Activate the Transparent Commands toolbar and select .

**2** Select the point in the drawing.

**3** If you are prompted, enter an elevation.

If the point you selected in the previous step has an elevation, it is listed as the default elevation, which you can select by pressing Enter. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

# Using a Point Name to Specify a Point Location

You can specify a point location by referencing the name of an existing point.

The system searches first in the active drawing, and then in the project, if applicable.

A transparent command can only be used within a running command.

**NOTE** Use the Prompt For Point Names setting to control whether points are created with point names. For more information, see Editing the Default Point Creation Settings (page 415).

**To specify a point location by point name**

1  When you are prompted to enter a point in a command, do one of the following:

   ■  Enter '**pa**.

   ■  Activate the Transparent Commands toolbar and select .

2  Enter a point name.

3  If you are prompted, enter an elevation.

   If the point you specified in the previous step has an elevation, it is listed as the default elevation, and you can select it by pressing Enter. For more information, see Entering an Elevation for a Transparent Command (page 1599).

   **NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

**Quick Reference**

Command Line

'PA

Toolbar

Transparent Commands toolbar ➤ 

# Using a Station Offset to Specify a Point Location

You can specify a point location using a station and an offset along an alignment.

An alignment must exist in the current drawing, and the specified station value must fall within the extents of the alignment.

A transparent command can only be used within a running command.

**To specify a point location by station offset**

1   When you are prompted to enter a point in a command, do one of the following:
    ■  Enter '**so**.

    ■  Activate the transparent command toolbar and select  .

2   Specify a station by either entering a station value or clicking in the drawing.

3   Specify a distance by either entering an offset or clicking in the drawing.
    The coordinates of the point are calculated.

4   Do one of the following:
    ■  Enter another station and follow the prompts to specify another location.

    ■  Press Esc to end the transparent command.

5   If you are prompted, enter an elevation. For more information, see Entering an Elevation for a Transparent Command (page 1599).

---

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

---

### Quick Reference

Command Line

  'SO

Toolbar

Transparent Command toolbar ➤ 

# Using a Profile Station and an Elevation to Specify a Point Location

You can specify the X and Y coordinates for a point by identifying a station along the parent alignment of the profile view.

Specify the alignment station and the point elevation either graphically or numerically.

A transparent command can only be used within a running command.

**To specify a point location using a station along the parent alignment of a profile view**

1   When you are prompted to enter a point in a command, do one of the following:
    ■  Enter '**stae**.

    ■  Activate the transparent command toolbar and select  .

2   If the main command is not a profile layout or profile view label command, select a profile view.

3   In plan view, select an alignment.

4   Specify a station on the alignment by either entering the station value or clicking a point on or near the alignment in plan view.
    A selection marker displays the current location on the alignment.

**5** Select the profile view again.

**6** Specify an elevation in the profile view by either entering an elevation value or clicking a point in the profile view.

In the profile view, a selection marker displays the current elevation. The station location is held at the station you selected.

**7** Do one of the following:

■ Enter another station and elevation.

■ Press Esc to end the transparent command.

### Quick Reference

Command Line

'STAE

Toolbar

Transparent Command toolbar ➤ 

# Using an Alignment and Surface to Specify a Point Location

You can specify the X and Y coordinates for a point by identifying a station along an alignment and location on a surface.

Define the profile layout entities by specifying a station and elevation based on plan view objects.

Elevation data can be derived from finished surfaces such as corridors and grading or from an existing surface.

A transparent command can only be used within a running command.

**To specify a point location in a profile view using an alignment and surface**

**1** When you are prompted to enter a point in a command, do one of the following:

■ Enter '**sse**.

■ Activate the transparent command toolbar and select  .

**2** In plan view, select the parent alignment of the profile view.

**3** Select the surface from which you want to derive elevation data.

**4** Select a point on the surface.

**5** Select the profile view again.

**6** Do one of the following:

■ Select additional locations on the surface to create additional points with the corresponding alignment station and elevation values.

■ Press Esc to end the transparent command.

# Using an Alignment and COGO Point to Specify a Point Location

You can specify the X and Y coordinates for a point by identifying an alignment and COGO point.

COGO points can be used to define a layout profile include points along an alignment, survey points, or corridor points.

Define profile layout entities by specifying a station and elevation based on plan view objects. Examples of COGO points that may be used to define a layout profile are points along the parent alignment or other alignment, survey points, or points derived from a corridor.

A transparent command can only be used within a running command.

**To specify a point location in a profile view using an alignment and a COGO point**

1  When you are prompted to enter a point in a command, do one of the following:
   - Enter '**spe**.

   - Activate the transparent command toolbar and select  .

2  In plan view, select an alignment.

3  Select a COGO point.

4  Select the profile view again.

5  Do one of the following:
   - Select additional points in plan view to create additional points with the corresponding alignment station and elevation values.

   - Press Esc to end the transparent command.

**Quick Reference**

Command Line

 'SPE

Toolbar

Transparent Command toolbar ➤

# Using Station and Elevation to Specify a Point Location in a Profile View

You can specify the X and Y coordinates for a point by identifying a station and an elevation within a profile view.

Specify the values within the profile view, either graphically or numerically.

A transparent command can only be used within a running command.

**To specify a point location in a profile view using station and elevation**

1   When you are prompted to enter a point in a command, do one of the following:
    - Enter '**pse**.

    - Activate the transparent command toolbar and select .

2   Select a profile view.

3   Specify a station in the profile view by either entering the station value or clicking a point in the profile view.

4   Specify an elevation in the profile view by either entering an elevation value or clicking a point in the profile view.

5   If you are working within a running command, do one of the following:
    - Enter another station and elevation.

    - Press Esc to end the transparent command.


## Quick Reference

Command Line

   'PSE

Toolbar

Transparent Command toolbar ➤ 


# Using Grade and Length to Specify a Point Location in a Profile View

You can specify a point location using grade and length values.

Specify the values within the profile view, either graphically or numerically.

At least one point must be entered in the current running command before you use this transparent command. A transparent command can only be used within a running command.

**To specify a point location in a profile view using grade and length**

1   When you are prompted to enter a point in a command, do one of the following:
    - Enter '**pgl** (the last letter is a lower case L).

    - Activate the transparent command toolbar and select .

**2** Select a profile view.

**3** Specify a grade by either entering a numeric value or clicking two points in the profile view.

**4** Specify a length, which is the horizontal distance from the last point picked in the running command, by either entering a value or clicking two points.

**5** If you are working within a running command, do one of the following:

■ Enter another grade and length.

■ Press Esc to end the transparent command.

### Quick Reference

Command Line

'PGL

Toolbar

Transparent Command toolbar ➤

# Using Grade and Station to Specify a Point Location in a Profile View

You can specify a point location using grade and station values.

Specify the values within the profile view, either graphically or numerically.

At least one point must be entered in the current running command before you use this transparent command. A transparent command can only be used within a running command.

**To specify a point location in a profile view using grade and station**

**1** When you are prompted to enter a point in a command, do one of the following:

■ Enter '**pgs**.

■ Activate the transparent command toolbar and select .

**2** Select a profile view.

**3** Specify a grade by either entering a numeric value or clicking two points in the profile view.

**4** Specify a station in the profile view by either entering a numeric value or clicking a point in the profile view.

**5** If you are working within a running command, do one of the following:

■ Enter another grade and station.

■ Press Esc to end the transparent command.

### Quick Reference

Command Line

'PGS

Toolbar

Transparent Command toolbar ➤ 

# Specifying a Length by Selecting an Object

Use the Match Length transparent command to specify a distance value as the length of an existing object.

A transparent command can only be used within a running command.

**To specify a length by selecting an object**

1 When you are prompted to enter a length or a distance in a command, do one of the following:
   - Enter '**ml** (the last letter is a lowercase L).

   - Activate the Transparent Commands toolbar and select  .

2 Select an AutoCAD line or a solved alignment line.


## Quick Reference

Command Line

   'ML

Toolbar

Transparent Commands toolbar ➤ 

# Specifying a Radius by Selecting an Object

Use the Match Radius transparent command to specify a radius value as the radius of an existing object.

A transparent command can only be used within a running command.

**To specify a radius by selecting an object**

1 When you are prompted to enter a point in a command, do one of the following:
   - Enter '**mr**.

   - Activate the Transparent Commands toolbar and select  .

2 Select an AutoCAD arc or a solved alignment curve.


## Quick Reference

Command Line

   'MR

Toolbar

Transparent Commands toolbar ➤

# Entering an Elevation for a Transparent Command

After you use a transparent command to establish the X and Y coordinates of a point location, you can specify the Z coordinate.

Specify an elevation using any of the following methods:

■ Accept the default elevation

■ Enter an elevation value

■ Specify a grade

■ Specify a slope

■ Take the elevation from the current surface

The default elevation, displayed at the prompt, is determined by the transparent command in use. If the current transparent command is 'PN (point number), 'PO (point object), or 'PA (point name), the default elevation is taken from the specified point. In all other cases, the default elevation is the AutoCAD default elevation.

A transparent command can only be used within a running command.

**NOTE** Elevation prompting for transparent commands is controlled by the Transparent Command setting Prompt For 3D Points. For more information, see Transparent Command Settings (page 1581).

**To enter an elevation for a transparent command**

1 Run a transparent command until you are prompted to enter an elevation.

2 Do one of the following:

■ To accept the default elevation that is displayed on the command line, press Enter.

■ Enter **g**. The elevation is calculated using the entered grade and a previously entered point.

■ Enter **s**. The elevation is calculated using the entered slope and a previously entered point.

■ Enter **y**. The elevation is taken from the X, Y location on the current surface.

■ Enter **t**. If there is more than one surface in the drawing, you are prompted to select the surface from which the elevation is read.

**NOTE** During a running command, the default prompts for elevation change, depending on how the previous elevation was entered. Enter **e** to reset the prompts back to the defaults described above.

# Using Point Filters Within Transparent Commands

Use Civil point filters to enter known point information when you are prompted for a point location within a transparent command.

A point filter offers you three ways to specify a point location:

■ enter a point number

■ enter northing and easting values

■ select a point in the active drawing

To activate a point filter, do one of the following:

■ Enter the point filter on the command line. You must always begin a transparent command point filter with a period (.).

■ Select the point filter from the transparent command filter toolbar. If the toolbar is not displayed, right click any toolbar and select Transparent Command Filters.

The following table lists the point filters available within the Civil transparent commands.

| To specify a location using this information... | Enter this point filter... | Or click this icon on the Transparent Command Filter Toolbar... |
| --- | --- | --- |
| point number | .p |  |
| a point in a drawing | .g |  |
| known northing and easting | .n |  |

A point filter is active only while you enter a single point location. Control then returns to the main transparent command.

Point filters are available for use only when you are prompted for a point location during the following transparent commands:

■ Angle Distance ('AD)

■ Bearing Distance ('BD)

■ Deflection Distance ('DD)

■ Azimuth Distance ('ZD)

■ Station Offset ('SO)

■ Side Shot ('SS)

When you are working with the .p (point number) point filter, the system searches for the point first within the active drawing, then within the project, if the drawing is attached to a project.

**To use a point filter within a transparent command**

1 When you are prompted for a coordinate location from within a transparent command, do one of the following:

■ To specify a point using a point number, enter **.p** and enter the point number.

■ To specify the coordinates from a point object in a drawing, enter **.g** and select the point.

■ To enter a known northing and easting, enter **.n** and enter the northing and easting values.

2 Continue with the transparent command.

# Transparent Command Reference

Use Transparent Command to quickly access functionality within a larger operation.

The following table lists the transparent commands that are used to specify point locations.

| To specify a point location using this information... | Use this transparent command... | Toolbar Icon |
|---|---|---|
| angle and distance (page 1582) | 'AD | |
| bearing and distance (page 1584) | 'BD | |
| azimuth and distance (page 1586) | 'ZD | |
| deflection angle and distance (page 1585) | 'DD | |
| northing and easting (page 1588) | 'NE | |
| grid northing and grid easting (page 1589) | 'GN | |
| latitude and longitude (page 1590) | 'LL | |
| point number (page 1590) | 'PN | |
| point name (alias) (page 1592) | 'PA | |
| a point in a drawing (page 1591) | 'PO | |
| side shot from a point (page 1587) | 'SS | |
| station and offset (page 1592) | 'SO | |
| station along the parent alignment of a profile view (page 1593) | 'STAE | |
| station along the parent alignment of a profile view, elevation from surface (page 1594) | 'SSE | |
| station along the parent alignment of a profile view, elevation from a COGO point (page 1595) | 'SPE | |
| station and elevation in a profile view (page 1596) | 'PSE | |
| grade and length in a profile view (page 1596) | 'PGL | |
| grade and station in a profile view (page 1597) | 'PGS | |

The following table lists the transparent commands that are not used to specify point locations:

| To specify this information when prompted... | Use this transparent command... | Toolbar Icon |
| --- | --- | --- |
| an object's length (page 1598) | 'ML | |
| an object's radius (page 1598) | 'MR | |

# Utilities

# 38

You can use the AutoCAD Civil Utilities to perform operations not related to a specific object or command.

## Coordinate Geometry (COGO)

Use coordinate geometry to enter accurate geometry when creating objects and find out accurate geometry for existing objects.

## Overview of Coordinate Geometry

Use coordinate geometry commands to input and extract locations in coordinate geometry.

Use the transparent commands to create objects using accurate geometry. For example, you can create parcel boundaries from legal documents or survey data. The COGO input commands expand the normal object creation commands by letting you use coordinate geometry to specify the points for the object.

Use the inquiry commands to get accurate geometric information about objects in your drawing. For example, you can determine the coordinates of a manhole or the centerline of a new road.

## Using Coordinate Geometry to Specify Points

Use the transparent input commands to create objects using accurate geometry. For example, you can create parcel boundaries from legal documents or survey data. For more information, see Transparent Commands (page 1581).

## Measuring Coordinate Geometry

Use the COGO inquiry commands to extract geometric information from objects such as lines, curves, closed polylines, and polygons. This information is useful if you want to verify the accuracy of your data.

**NOTE** The COGO inquiry commands use the World Coordinate System (WCS) and ignore current User Coordinate System (UCS) settings. Therefore, north is always considered to point along the WCS positive Y axis, and inquiry results are reported in WCS coordinates.

## Adding Distances

You can calculate the total of several disjunct distances on your drawing.

Specify the distance by selecting points in your drawing, enter distances on the command line, or select numeric text, such as measurements, in your drawing.



Select all the distances you want to add. When you press Enter, AutoCAD Civil displays the total of all the distances.

**To add distances**

1   Click Analyze tab ➤ Inquiry panel ➤ Add Distances $\Sigma_{\blacksquare}$ .

2   Specify the first distance by doing one of the following:

   ■   Enter the distance on the command line.

   ■   Select two locations in the drawing.

   ■   Enter s. Select numeric text in the drawing, such as a measurement.

3   Enter as many additional distances as you want.

4   When you finish selecting distances, press Enter to view the total of the distances.


### Quick Reference

Ribbon

Analyze tab ➤ Inquiry panel ➤ Add Distances $\Sigma_{\blacksquare}$

Menu

Inquiry menu ➤ Add Distances

Command Line

CgADist

## Displaying Continuous Distance

You can add and display the distance between one point and a number of other points, or between a series of points.

The Base option measures the distance from the starting point to each of the points you select.



The Continuous option measures the distance from the starting point to the next point in a continuous line.



**TIP**   To add distances that are not continuous, see Adding Distances (page 1604).

**To display the distance between points**

1   Click Analyze tab ➤ Inquiry panel ➤ Continuous Distance ⇔ .

2   Select Base or Continuous.

   ■   Base: Always measures the distance from the first point you select to each of the additional points you select.

   ■   Continuous: Measures the distance from one point to the next.

3   Select two points to display the distance between the points.

4   Select another point. If you selected Base, AutoCAD Civil displays the distance from first (or base) point to the new point. If you selected Continuous, AutoCAD Civil displays the distance from the last point to the new point.

5   When you finish selecting points, press Enter to view the total of the distances.

### Quick Reference

## Displaying Angle Information

You can display the acute and obtuse angle between points or intersecting lines.

This command uses the Angle settings as specified on the Ambient tab (page 68) in the Drawing Settings dialog box.

**To display angle information**

1 Click Analyze tab ➤ Inquiry panel ➤ Angle Information ◹⃰ .

2 Select two lines, or enter p to specify points.

3 If you entered p, specify a starting point, a vertex, and an ending point.

### Quick Reference

## Displaying COGO Information for Lines and Arcs

You can display coordinate geometry for lines and arcs. This information includes line and curve details, area, and coordinates.

This command uses the Angle, Distance, and Direction settings as specified on the Ambient tab (page 68) in the Drawing Settings dialog box.

**To display COGO information**

1 Click Analyze tab ➤ Inquiry panel ➤ Line And Arc Information ⌐ .

2 Select the line or arc, or enter p to specify the points for a line.

3 If you entered p, specify a starting point and an ending point for the line.

### Quick Reference

## Listing the Slope or Grade between Points

You can display the slope, grade, and horizontal distance between two points.

This command uses the Elevation, Distance, and Grade/Slope settings as specified on the Ambient tab (page 68) in the Drawing Settings dialog box.

**To display the slope between two points**

1  Click Analyze tab ➤ Inquiry panel ➤ List Slope .

2  Select a line or an arc, or enter p to specify points.

3  If you entered p, specify a starting point and an ending point for the line.

### Quick Reference

# Mapcheck Analysis

Perform a Mapcheck Analysis by selecting AutoCAD Civil line and curve labels to determine values from label objects based on the precision of the annotation of the label object, or enter mapcheck data manually.

A mapcheck analysis is typically performed for each labeled polygon that represents a parcel, a labeled parcel object, or survey figure to ensure that errors and omissions are minimized. If error is introduced and accumulated, the coordinates of the last segment endpoint will not equal that of the point of beginning (POB) which creates the 'error of closure'.

The report is computed based on the start and end coordinates of each parcel segment, relative to the coordinates of the POB and the previous segment.

In the Survey Command Window you have the option to list mapcheck information that computes the error of closure for a figure based on the precision settings for direction and distance values which you specify in the  Survey Database settings (page 2378) For more information, see Listing Mapcheck Information for a Figure (page 356).

In the Parcel Properties you can specify Inverse Analysis and Mapcheck Analysis. For more information, see Exporting a Parcel Inverse or Mapcheck Report (page 830).

# Performing a Mapcheck Analysis

Use the Mapcheck Analysis command to perform a mapcheck that is based on the precision of the annotation of the label object.

The Mapcheck Analysis command computes mapcheck calculations directly from the selected labels. To perform a mapcheck you must specify a point of beginning (POB) for each parcel. For each segment within the parcel, you select a label containing a direction, direction and distance, distance label, or curve label until the requirements to compute the next vertex of the current segment are met.

As you perform a mapcheck, temporary graphics are displayed in the drawing to indicate the sides that have been entered for each parcel. The current side is displayed in bold indicating origin and direction for that side. The temporary graphics also provide a visual confirmation of possible errors, or indicate that the direction of a line or curve needs to be flipped or reversed.

**NOTE** The mapcheck report data is stored in the drawing database so you can continue a mapcheck from a previous AutoCAD Civil session. The Mapcheck Analysis window must be open to enable the display of the temporary graphics.

From the POB, the annotation for each side in the parcel is used to calculate the coordinates of the endpoint of the side. As you enter data for each parcel, the Output View in the Mapcheck Analysis window updates to indicate error of closure and area based on the previously entered sides. Entering the information for the last side closes to the POB. The Mapcheck Analysis output view contains the following information:

■ POB coordinates

■ A sequential list of each course, direction and distance, or curve data, and end point coordinates

■ An error of closure summary: Error of closure: 1 part in <X parts>

■ Error of closure direction: <bearing or azimuth>

■ Error of closure distance: <distance>

■ Perimeter distance: <total distance around the parcel>

■ Area: <area units (acres/hectares, and square feet/meters>

For each parcel or survey figure, the area annotation is compared to the mapcheck area. If the error of closure is acceptable, for example a closure that exceeds one part in 10,000 parts and the area annotation matches, the annotations are considered to be mathematically correct.

📖 Tutorial Exercise: Performing a Map Check Analysis with Parcel Labels

**To perform mapcheck analysis from AutoCAD Civil label objects**

1  Click Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck 🗹 .

2  In the Mapcheck Analysis window, click 📄 New Mapcheck Report.

3  Enter a name for the Mapcheck, for example LOT 1.

4  Click 🖥 to Use Command Line Interface and follow the prompts at the command line.

5  Specify a Point of Beginning. The interactive graphics display the POB in the drawing.

---
**NOTE** Colors displayed in the temporary graphics are controlled by the Mapcheck command settings. For more information, see Mapcheck Command Default Settings (page 1615).

---

The following illustration shows the Point of Beginning:



6  At the command line you are prompted to select a label. After selecting a label, the interactive graphics display in the drawing. If you need to make a change, at the command line you have options to do the following:

- Clear: Clears the current side.

- New: Starts a new mapcheck report.

- Flip: Changes the direction of the curve to either clockwise or counter-clockwise.

- Reverse: Changes the direction of the line or curve.

---
**NOTE**  If the mapcheck requires more information to make the calculation, a red circle displays in the drawing and you are prompted at the command line.

---

The following illustration shows how the Reverse command can be used to change the direction of the mapcheck line:

The following illustration shows an example of using the Flip command:

**7** Continue to follow the command line prompts by selecting labels until you return to the Point of Beginning.

The following illustration shows the continuation of the mapcheck towards the POB:

**8** Click ![icon] New Mapcheck to add a new mapcheck to the existing Mapcheck Report. Repeat steps 5 through 7.

**9** Click ![icon] Output View to review the mapcheck analysis report. Click ![icon] to return to the input window.

**10** Optionally, in either the Input View or the Output View, click ![icon] Insert Mtext to insert either the current mapcheck or all mapchecks into the drawing. Specify an insertion point in the drawing and insert the Mapcheck Report.

### Quick Reference

Ribbon

Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck ![icon]

Menu

General menu ➤ Mapcheck Analysis ![icon]

Command Line

Mapcheck

Dialog box

Mapcheck Analysis Window (page 2447)

## Manually Entering Mapcheck Data

Create a mapcheck report by manually entering data.

You can enter data from a paper drawing into the Mapcheck Analysis window to create a mapcheck report. For each mapcheck, you can create a polyline which can then be used to create AutoCAD Civil objects such as a survey figure.

![icon] Tutorial Exercise: Performing a Map Check Analysis by Manually Entering Data

**To create a mapcheck report by manually entering data**

---

**NOTE** To manually enter mapcheck data, ensure that the 🖳 Use Command Line Interface is toggled off.

---

**1** Click Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck ✅.

**2** In the Mapcheck Analysis window, click 🗋 New Mapcheck Report.

**3** Enter a name for the Mapcheck, for example LOT 1.

**4** Enter the Easting and Northing values for the Point of Beginning or click ⬛ and select a point in the drawing.

**5** Click ⬛ to create a New Side. Specify the values for the side.

For a line side, enter the following:

■ Side Type: Select Line from the drop-down list.

■ Angle Type: Select Direction, Angle, or Deflection Angle from the drop-down list.

■ Angle: Enter the angle value.

■ Reverse Direction: Specify Yes or No.

■ Distance: Enter the distance value.

For a curve side, enter the following:

■ Side Type: Select Curve from the drop-down list.

■ Curve Direction: Select Clockwise or Counter-clockwise from the drop-down list.

■ Traverse Method: Select Across Chord or Through Radius from the drop-down list.

> **NOTE** This option is used to specify how the error of closure is calculated and displayed in the Output View. If you select Across Chord, the endpoint of the curve is calculated from the Chord direction and distance. If you select Through Radius, the endpoint of the curve is calculated based on the direction to the radius point, the radius length, the delta angle, and the direction from the radius point to the point of tangency. If this option is selected, additional information is displayed in the Output View.

■ Radius: Specify the radius value.

■ Arc Length: Specify the arc length.

■ Delta Angle: Specify the delta angle.

■ Tangent: Specify a tangent.

■ Angle Type: Select Direction, Angle, or Deflection Angle from the drop-down list.

■ Chord Angle: Specify a chord angle.

■ Reverse Direction: Specify Yes or No.

■ Chord Distance: Specify the chord distance.

**6** Click ⬅ Insert Side Before, or ➡ Insert Side After, if you have omitted a side before or after the current selected side in the mapcheck. Specify values as shown in the previous step.

**7** Click ✅ New Mapcheck to add a new mapcheck to the mapcheck report. Repeat steps 3 through 6.

**8** When you have completed entering data, click  Output View to view the mapcheck analysis. Click  Input View to return to the input window.

**NOTE** You can switch between input and output views at any time.

**9** Optionally, click  Create Polyline to create a polyline for the current mapcheck. The polylines can be converted to AutoCAD Civil objects.

**NOTE** If there is no current mapcheck then a polyline for all mapchecks are inserted into the drawing. Press ESC to clear a selection in tree view.

**10** Optionally, click  Copy To Clipboard to copy the contents of the input view or output view to the Windows Clipboard of the current mapcheck.

**NOTE** If there is no current mapcheck then all mapchecks are copied to the clipboard. Press ESC to clear a selection in tree view.

**11** Optionally, in either the Input View or the Output View, click  Insert Mtext. Specify an insertion point in the drawing and insert the mapcheck analysis report.

### Quick Reference

Ribbon

Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck 

Menu

General  menu ➤ Mapcheck Analysis 

Command Line

Mapcheck

Dialog box

Mapcheck Analysis Window (page 2447)

## Adjusting a Mapcheck Analysis

After creating a mapcheck report by manually entering data and determining that there is a reasonable error of closure, you can adjust the mapcheck to distribute the total error among the sides to precisely match the point of beginning and endpoint coordinates.

**To adjust a Mapcheck**

**1** In the Mapcheck Analysis window, right-click a Mapcheck Report  ➤  Adjust Mapcheck.

**2** In the Mapcheck Analysis dialog box, specify the Horizontal Adjustment Method, Horizontal Closure Limit, and Angle Error Per Set. For more information, see Adjust Mapcheck Analysis Dialog Box (page 2450).

**3** Specify Yes to update the mapcheck. The  icon is displayed next to the mapcheck name indicating the mapcheck has been adjusted. To clear the adjustment, right-click the Mapcheck ➤ Clear Adjustment.

### Quick Reference

Mapcheck Analysis Window

  Input View ➤ right-click Mapcheck ➤ Adjust Mapcheck

Dialog Box

  Adjust Mapcheck Analysis Dialog Box (page 2450)

# Inserting a Mapcheck Report as Mtext

You can insert a Mapcheck report as an Mtext object in the drawing.

📖 Tutorial Exercise: Working with Mapcheck Data

**To insert a Mapcheck Report into the drawing**

1   Click Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck ☑.

2   In either the Input View or the Output View of the Mapcheck Analysis window, click ⌀ A₊ Insert Mtext.

### Quick Reference

Ribbon

  Analyze tab ➤ Ground Data panel ➤ Survey drop-down ➤ Mapcheck ☑

Menu

  General menu ➤ Mapcheck Analysis ☑

Command Line

  Mapcheck

Dialog box

  Mapcheck Analysis Window (page 2447)

# Mapcheck Command Default Settings

Change the Mapcheck default command settings in the General collection on the Settings tab in Toolspace

**To change Mapcheck command settings**

1   On the Settings tab in Toolspace, click General ➤ Commands ➤ MapCheck.

2   In the Edit Command Settings-MapCheck dialog box, expand the Mapcheck group and specify values for the mapcheck properties.

### Quick Reference

Toolspace

  Settings tab ➤ General collection ➤ Commands ➤ MapCheck

# Attaching Notes

With the Notes command, you can add either detailed or reference information to a selected object.

Use the Notes command to write textual information that can be viewed with the object and associate a separate reference file (document, spreadsheet, image or photo, and so on).

View the notes and documents later by using the Extended Data tab of the AutoCAD Properties Palette.

# Attaching Text to an Object

You can attach a text file to any AutoCAD object.

**To add text notes to a drawing**

1   At the Command prompt, enter Notes.

2   In the drawing, select an object on which you want to place a note.

3   Press Enter.
    The Notes dialog box is displayed.

4   Click the Notes tab.

5   In the text box, enter the desired text.

6   Click OK to attach the text file to your drawing.

### Quick Reference

Ribbon

  Not available on the ribbon

Menu

  General menu ➤ Utilities ➤ Notes

Dialog Box

  Notes (page 2451)

Command Line

  Notes

# Attaching External Reference Documents to an Object

Attach, edit, and delete external reference documents to any AutoCAD object in a drawing.

**To attach, edit, or delete an external reference document**

1   At the Command prompt, enter Notes.

2   In the drawing, select an object on which you want to attach an external reference document.

3   Press Enter.
    The Notes dialog box is displayed.

**4** To attach, edit, or detach a reference file, click the Reference Docs tab, and do any of the following:

- To attach a reference file: Click Add, select a document in the Select Reference Document dialog box, and click Open. You can type a description of the reference file on the Reference Docs tab.

- To edit a reference file: Select the file name in the list, click Edit, and change the document or the description in the Reference Document dialog box. To edit the file itself, double-click the reference file name to start its application.

- To detach a reference file: Select the file name in the list and click Delete.

**5** Click OK.

## Quick Reference

Ribbon

Not available on the ribbon

Menu

General menu ➤ Utilities ➤ Notes

Dialog Box

Notes (page 2451)

Command Line

Notes

# Using the Object Viewer

The Object Viewer displays objects you select in your drawing, based on the current view that is set in the drawing.

Civil objects are displayed in the Object Viewer using the Plan or Model settings specified in their styles. For example, in a 3D view, objects are displayed using their Model settings.

You can manipulate the viewing angle and then set the drawing view equal to the view in the Object Viewer by clicking ⊕ .

The Object Viewer contains the ViewCube and the SteeringWheel. For more information, see Use ViewCube and Navigation Wheels in the AutoCAD Help.

**To display objects from your drawing in the Object Viewer**

**1** Click View tab ➤ Palettes panel ➤ Object Viewer 🔍 .

**2** Select the objects you want to view.

**3** Press Enter.

The Object Viewer is displayed.

**4** From the View Control list at the upper right of the Object Viewer, select any of the following.

- **Top, Bottom, Left, Right, Front, or Back:** Sets the current view to the selected view.

- **SW Isometric, SE Isometric, NE Isometric, or NW Isometric:** Sets the current view to the selected isometric view.

**5** From the Visual Style list, select one of the following object appearance styles.

- **3D Hidden** To display objects in the current view in a 3D wireframe representation with all visible lines displayed as continuous and all hidden lines displayed as dashed.

- **3D Wireframe** To display objects in the current view in a 3D wireframe representation with all lines displayed, including those objects hidden by other objects.

- **Conceptual** To display objects in a transparently shaded 3D view with all lines visible.

- **Realistic** To display objects in a shaded 3D view.

**6** Choose from the following zoom options from the Object Viewer to increase or decrease the apparent size of objects in the current viewport:



**7** Click  to set the view in the drawing to match the current view in the Object Viewer.

> **NOTE** AutoCAD Civil supports only the Standard Display Configuration.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ➤ Object Viewer 

Menu

General menu ➤ Utilities ➤ Object Viewer

Command Line

ObjectViewer

Dialog Box

Object Viewer (page 2452)

# Performing Inquiries on AutoCAD Civil Objects

Use the Inquiry Tool to run inquiries on AutoCAD Civil objects. You can extract information from the design model to aid in design, validation, and analysis. See Inquiry Types (page 1620) for a list of available inquiries.

The Inquiry Tool is in a window you can keep open on screen while you do other work. Like the Toolspace, you can dock the window or turn on the AutoHide feature so it displays only when you move your mouse over it to save desktop space.

The Inquiry Tool consolidates and replaces some of the listing and inquiry commands that were in previous releases of AutoCAD Civil.

For more information, see Inquiry Tool Dialog Box (page 2462).

# Performing an Inquiry

Use the Inquiry Tool to run a selected inquiry on drawing objects.

Choose an inquiry type and follow prompts to select the data. After the inquiry completes, you can copy the data to another file such as a Microsoft© Excel spreadsheet.

There are two ways to select the data for an inquiry. When you first select the inquiry to run, you are prompted to select the data required for the inquiry. If there is only one object of that type in the drawing, it is selected automatically. After the inquiry is complete, you can use the object select buttons. Click  or  in the Inquiry Tool window to change the selection of data.

You can also run an inquiry by selecting an object in the drawing and accessing the Inquiry command from the shortcut menu. In this case you are not prompted for the main object, but you are prompted for any additional object input the inquiry requires.

**To perform an inquiry**

1   Click Analyze tab ➤ Inquiry panel ➤ Inquiry Tool  .

2   In the Inquiry Tool window, under Select An Inquiry Type, click the Down arrow and select the inquiry to run.

   After you select an inquiry type, you are prompted to select the objects that are required for the inquiry.

   ---
   **NOTE** To run Corridor queries, a corridor section view must be active. Select a corridor. Click Corridor tab ➤ Modify panel ➤ Corridor Section Editor  . Then select the station to query. Use the Zoom command to zoom closer to the view.
   ---

3   Select the objects and locations for the inquiry. You can use Osnaps and transparent commands to select the objects.

   The data is displayed in the Inquiry Tool window.

4   To change selected objects or locations, click  or  in the Inquiry Tool window next to a value and select the new object or location in the drawing.

## Quick Reference

Ribbon

   Analyze tab ➤ Inquiry panel ➤ Inquiry Tool 

Menu

   General menu ➤ Inquiry Tool

Object Shortcut Menu

   Right-click object ➤ Inquiry. The most recently used inquiry type for the selected object runs automatically.

Command Line

   ShowInquiry
   HideInquiry
   DockInquiry
   FloatInquiry

# Copying Inquiry Data

After you run an inquiry, you can copy the data to the clipboard, or you can send the inquiry data to the AutoCAD text window and copy the data from there.

The option to copy the inquiry to the AutoCAD text window continually sends the data to the text window as the inquiry proceeds.

**To copy inquiry data to the clipboard**

➤   Right-click in the Inquiry Tool results area and click Copy To Clipboard. Or click 📋 .

**To copy an individual value to the clipboard**

1   Right-click the value in the Inquiry Tool.

2   Click Copy Value To Clipboard.

**To send a running copy of inquiry data to the AutoCAD text window**

1   Click 📋 Copy To Text Screen.

As you run the inquiry, the data is sent to the text window.

2   Press F2 to expand the text window and view the data.

### Quick Reference

Ribbon

Analyze tab ➤ Inquiry panel ➤ Inquiry Tool 📇

Menu

General menu ➤ Inquiry Tool

Command Line

ShowInquiry

# Inquiry Types

You can run inquiries on points, surfaces, alignments, profiles, profile views, sections, section views, and corridor sections.

| Inquiry Type | Inquiry Input | Inquiry Results |
|---|---|---|
| Point Inverse | ■   Two points | ■   Direction |
| | | ■   Horizontal Distance |
| | | ■   Grid Direction |
| | | ■   Grid Distance |
| Surface Elevation and Grade At Point | ■   Surface | ■   Surface elevation at point |
| | ■   Point on surface | ■   Grade (percentage) |
| | | ■   Slope |

| Inquiry Type | Inquiry Input | Inquiry Results |
|---|---|---|
| | | ■ Grade (degree) |
| | | ■ Slope direction |
| Surface Elevation and Grade Between Points | ■ Surface<br>■ Two points on surface | ■ Surface elevation at each point<br>■ Elevation difference between points<br>■ Distance between points<br>■ Grade (percentage)<br>■ Slope<br>■ Grade (degree)<br>■ Slope direction |
| Alignment Station and Offset at Point | ■ Alignment<br>■ Offset point | ■ Station at point<br>■ Offset at point |
| Alignment Station, Offset, and Profile Elevation at Point | ■ Alignment<br>■ Profile<br>■ Horizontal offset point | ■ Station at point<br>■ Offset at point<br>■ Profile elevation at alignment station<br><br>NOTE This command lists the elevation not at the offset point, but on the alignment. Use the following query to list the elevation at the selected offset point. |
| Alignment Station, Offset, and Surface Elevation at Point | ■ Alignment<br>■ Surface<br>■ Horizontal offset point | ■ Station at point<br>■ Offset at point<br>■ Surface elevation at point |
| Alignment Two Stations and Offsets at Point | ■ Two alignments<br>■ Offset point | ■ Stations and offsets (in relation to both alignments) |
| Profile View Station and Elevation at Point | ■ Profile View<br>■ Point on profile view | ■ Station at point<br>■ Elevation at point |
| Profile View Elevation and Grade Between Points | ■ Profile View<br>■ Two points on profile view | ■ Elevation difference between points<br>■ Distance between points<br>■ Grade (percentage) |

| Inquiry Type | Inquiry Input | Inquiry Results |
|---|---|---|
| | | ■ Slope |
| | | ■ Grade (degree) |
| Profile Station and Elevation at Point | ■ Profile<br>■ Point on profile | ■ Station at point<br>■ Elevation at point |
| Profile Elevation Difference at Station | ■ Profile view<br>■ Two profiles<br>■ Point on profile view | ■ Station at point<br>■ Elevation difference between profiles at point |
| Section View Offset and Elevation at Point | ■ Section view<br>■ Point on section view | ■ Station at point<br>■ Horizontal offset at point<br>■ Elevation at point |
| Section View Elevation and Grade Between Points | ■ Section view<br>■ Two points on section view | ■ Station of selected section view<br>■ Elevation difference between points<br>■ Distance between points<br>■ Grade (percentage)<br>■ Slope<br>■ Grade (degree) |
| Section Offset and Elevation at Point | ■ Section<br>■ Point on section | ■ Offset at selected point<br>■ Elevation of section at offset point |
| Section Difference at Offset | ■ Section View<br>■ Two sections<br>■ Offset point | ■ Offset at selected point<br>■ Elevation difference between the sections at offset point |

**TIP** To select a section, zoom into a section view and click the section object inside the view. If you sampled multiple surfaces and you have trouble selecting the correct section surface, edit the section view properties (Sections Tab (Section View Properties Dialog Box) (page 2282)) and turn off the display of the sections you do not want to select.

| | | |
|---|---|---|
| Corridor Section Offset and Elevation at Point | ■ **Prerequisite**: Run View/Edit Corridor Sections Command and select station<br>■ Offset/Elevation point | ■ Offset at selected point<br>■ Elevation at selected point |

| Inquiry Type | Inquiry Input | Inquiry Results |
|---|---|---|
| Corridor Section Elevation and Grade Between Points | ■ **Prerequisite**: Run View/Edit Corridor Sections Command and select station<br><br>■ Two Offset/Elevation points | ■ Elevation difference between points<br><br>■ Distance between points<br><br>■ Grade (percentage)<br><br>■ Slope<br><br>■ Grade (degree) |
| Corridor Section Elevation Difference at Offset | ■ **Prerequisite**: Run View/Edit Corridor Sections Command and select station<br><br>■ Two corridor surfaces<br><br>■ Offset/Elevation point | ■ Offset at selected point<br><br>■ Elevation difference between corridor surfaces at selected point |

# The Coordinate Tracker

You can use the Coordinate Tracker to view and capture X, Y, and Z drawing coordinates.

The Coordinate Tracker displays the X and Y coordinates of the cursor in the drawing. If the cursor is within the boundary of a specified surface, the surface elevation at the location of the cursor is displayed as the Z coordinate.

You can send the coordinate information to the command line for use in the active command, or you can copy the information for use in another application.

Because the Coordinate Tracker is an AutoCAD palette, you can use Auto-hide to minimize the display space it requires. You can also dock, move, and resize it. For more information about Auto-hide and AutoCAD palettes, see Working with AutoCAD Civil Windows (page 78).

# Viewing Cursor Coordinates Using the Coordinate Tracker

Use the Coordinate Tracker to view the drawing X and Y coordinates of the cursor and a Z coordinate value read from a specified surface.

**To view cursor coordinates using the Coordinate Tracker**

1 Click View tab ➤ Palettes panel ➤ Coordinate Tracker $\overset{\text{XYZ}}{\text{Q}}$ .

2 In the Coordinate Tracker (page 2462), in the Surface list, select the surface from which Z values (elevations) are read.

 If the drawing contains only one surface, it is automatically selected.

3 Move the cursor within the boundary of the selected surface.

 The drawing X and Y coordinates are updated from the cursor position within the drawing. The Z coordinate value is read from the specified surface when the cursor is within the surface boundary. If you move the cursor outside of the surface boundary, the Z value is not shown.

4 To suspend cursor tracking and capture the current X, Y, and Z coordinate values, do the following:

 ■ Enter the keys listed in the Coordinate Capture Keys edit box by pressing, and not releasing, each key in the specified order until you are holding all the specified keys down.

■ Release all keys.

Coordinate tracking is suspended and the current X, Y, and Z coordinates are captured.

5 Optionally, send the X, Y, and Z coordinates to the command line for use in the current command. You can edit the values before you send them. For more information, see Sending Coordinate Tracker Values to the Current Command (page 1624).

6 To resume cursor tracking, click ⇨ .

7 To hide the Coordinate Tracker, click ✕ .

### Quick Reference

Ribbon

View tab ➤ Palettes panel ➤ Coordinate Tracker <sup>XYZ</sup>⊙

Menu

General menu ➤ Coordinate Tracker

Command Line

ShowCT

Dialog Box

Coordinate Tracker (page 2462)

## Sending Coordinate Tracker Values to the Current Command

Use the Coordinate Tracker to capture and send drawing X, Y, and Z coordinate values to the current command.

You can send any of the following:

■ X and Y values only

■ Z (elevation) value only

■ X, Y, and Z values

Send the value required by the current command. If the current command requires a 3D (X, Y, and Z) value, an error occurs if you send a 2D (X and Y only) value.

Send a value by first moving the cursor to the desired location and entering the coordinate capture keys. This suspends tracking and captures the X, Y, and Z values. You can edit one or more of the values before you send them.

You send the values to the command line using the buttons directly beneath the X, Y, and Z values. The buttons are shown in the following illustration:



**TIP** Move the cursor over each button to view a tooltip that tells you what value the button sends to the command line.

**To send Coordinate Tracker values to the current command**

**1** Display the Coordinate Tracker. Select a surface if required. For more information, see Viewing Cursor Coordinates Using the Coordinate Tracker (page 1623).

**2** Run any command until you are prompted at the command line for a 2D point, a 3D point, or an elevation.

**3** Move the cursor until the Coordinate Tracker displays the desired coordinate and elevation values.

**4** To suspend cursor tracking and capture the current X, Y, and Z coordinate values, do the following:

- Enter the keys listed in the Coordinate Capture Keys edit box by pressing, and not releasing, each key in the specified order until you are holding all the specified keys down.

- Release all keys.

Coordinate tracking is suspended and the current X, Y, and Z coordinates are captured.

**5** Optionally, edit the coordinate information before sending it. In the Coordinate Tracker (page 2462), click in either the X:, Y:, or Z: edit box and enter a new value.

---

**TIP** Move the cursor over each button to view a tooltip that tells you what value the button sends to the command line.

---

**6** Do one of the following:

- To send the X and Y values, click [_____] .

- To send the Z value only, click [_____] .

- To send X, Y, and Z values, click [_____] .

**7** After you send a value, cursor tracking is automatically resumed. To resume cursor tracking without sending a value, click ⇨ .

## Quick Reference

Ribbon

View tab ➤ Palettes panel ➤ Coordinate Tracker ˣʸᶻ

Menu

General menu ➤ Coordinate Tracker

Command Line

ShowCT

Dialog Box

Coordinate Tracker (page 2462)

# Changing the Coordinate Tracker Coordinate Capture Keys

Change the capture key sequence that suspends coordinate tracking and captures the current X, Y, and Z values.

The *coordinate capture keys* are the keys you press to capture the current X, Y, and Z drawing coordinates. Then you can either send the values to the command line or copy them for use in another application. For more information, see Sending Coordinate Tracker Values to the Current Command (page 1624).

The capture key sequence always consists of any combination of Ctrl, followed by Alt and/or Shift, followed by a single character key. The following are examples of valid capture key combinations:

- *Ctrl+Shift+Z*
- *Ctrl+Alt+Z*
- *Ctrl+Shift+Alt+Z*

The default coordinate capture key sequence is *Ctrl+Shift+Z*.

You change the coordinate capture keys by clicking ⌨ and entering the new key sequence.

**To change the Coordinate Tracker coordinate capture keys**

1 Click View tab ➤ Palettes panel ➤ Coordinate Tracker.

2 In the Coordinate Tracker (page 2462), click ⌨ .

3 Press and hold down Ctrl.

4 While holding down Ctrl, press and hold down Alt and/or Shift.

5 While holding down the other keys, press and hold down a single character key.

6 Release all keys.

The Coordinate Capture Keys edit box displays the sequence of keys that you entered. If the Coordinate Capture Keys edit box is not updated, the sequence of keys that you entered is not valid. This can occur if the sequence does not begin with Ctrl followed by either Shift or Alt, or if the sequence is reserved for another function on your computer.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ➤ Coordinate Tracker ᴿˢ

Menu

General menu ➤ Coordinate Tracker

Command Line

ShowCT

Dialog Box

Coordinate Tracker (page 2462)

# The Event Viewer Vista

You can use the AutoCAD Civil Event Viewer to view event messages that are logged during an AutoCAD Civil drawing session.

The display of the Event Viewer can be controlled by changing the Show Event Viewer setting in the drawing's Ambient Settings (page 1856).

The Event Viewer is a Panorama vista. Each drawing can have a different Event Viewer session associated with it. When you change the active drawing, the Event Viewer session changes. If you do not activate the Event Viewer for a drawing, the Event Viewer is not displayed when that drawing becomes active. For information about Panorama vistas, see The Panorama Window (page 102).

The Event Viewer displays the *event tree* in the left pane and the *event list* in the right pane.

You use the event tree to control the events that are listed in the event list. For example, if you click the top collection in the tree, Event Viewer, all events are listed in the event list. If you expand the event tree and click either the Event Subsystem collection or Surfaces collection, only the events that were logged by either the Event Subsystem (which is the Event Viewer) or the Surfaces feature are listed in the event list.

When you first open the Event Viewer, the event list is empty if no events have been logged to the Event Viewer. When an event is logged to the Event Viewer, the appropriate collection is created, and the event is added to the event list.

You can save the event list to a log file as a permanent record, and open a previously created log file to view it in the Event Viewer. You can also export an event list to a comma-delimited or tab-delimited text file.

# Customizing the Event Viewer

You can control the content and appearance of the Event Viewer, including the columns displayed in the event list and the events listed there.

## Controlling the Columns Displayed in the Event Viewer

Control the columns that are displayed in the event list and the order in which they are displayed.

Columns that you can add or remove include Date, Time, Source, User, and Description.

You cannot remove the Type column from the display.

**To control the columns displayed in the Event Viewer**

1 Click View tab ➤ Palettes panel ▼ Event Viewer ⟁ .

2 To change the location of a column in the event list, in the Event Viewer Vista (page 2455), click and drag the heading right or left.

3 To add columns or remove columns, do the following:
   ■ In the Event Viewer, on the View menu, click Choose Columns.

   ■ In the Choose Columns (page 2459) dialog box, specify columns to be added or removed.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ▼ Event Viewer ⟁

Event Viewer Menu

View ➤ Choose Columns

Dialog Box

Choose Columns (page 2459)

## Controlling the Events Displayed in the Event Viewer

Control which events are displayed in the event list and the order in which they are displayed.

When you change the display of the event list by filtering, the events are hidden, but they are not deleted from the event list. However, when you use the Clear All Events command, the events are permanently deleted from the list.

For information about changing the columns that are displayed in the event list, see Controlling the Columns Displayed in the Event Viewer (page 1627).

For information about locating an event that matches specified criteria, see Finding a Specific Event in the Event Viewer (page 1630).

**To control the events displayed in the Event Viewer**

1  Click View tab ➤ Palettes panel ▼ Event Viewer ◁ᐞ .

2  In the Event Viewer (page 2455), do one of the following in the event tree:
   - Click the Event Viewer collection to display all the events in the event list.
   - Expand the Event Viewer collection and click a collection to display the events in that collection.

3  Click View menu ➤ Filter. Use the Filter Events (page 2457) dialog box to filter the event list to meet only specified criteria.

4  To display the event list by date, on the View menu, click one of the following:
   - Newest First: Displays the event list with the most recent event at the top.
   - Oldest First: Displays the event list with the least recent event at the top.

5  To sort the event list using a column heading, click the column heading.

6  Click Action menu ➤ Clear All Events to permanently remove all events from the event list.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ▼ Event Viewer ◁ᐞ

Event Viewer Menu

View ➤ Filter
Action ➤ Clear All Events

Dialog Box

Filter Events (page 2457)

## Working With Log Files in the Event Viewer

Open and create log files that contain event lists.

You can save the current event list to a binary log file, and also open an existing log file to view it.

For information about saving an event list to a comma-delimited or tab-delimited text file, see Exporting an Event List to a Text File (page 1630).

## Creating a Log File in the Event Viewer

Use the Save Log File As command to create a log file from the event list.

When you create a log file, all the events in the current event viewer session at the time the file is created are written to the file. Events that have been logged during the session, but that are not displayed in the event list due to filtering, are also written to the file. For more information about using filters to limit the events displayed in the event list, see Controlling the Events Displayed in the Event Viewer (page 1628).

**To create a log file in the Event Viewer**

1 Click View tab ➤ Palettes panel ▼ Event Viewer ⬠ .

2 In the Event Viewer (page 2455), on the Action menu, click Save Log File As.

3 In the Save As dialog box, browse to the folder where you want the log file to be located, enter a file name, and click Save.

The current event list is saved to the log file.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ▼ Event Viewer ⬠

Event Viewer Menu

Action ➤ Save Log File As

## Opening a Log File in the Event Viewer

Use the Open Log File command to open an existing Event Viewer log file.

**To open a log file in the Event Viewer**

1 Click View tab ➤ Palettes panel ▼ Event Viewer ⬠ .

2 In the Event Viewer (page 2455), on the Action menu, click Open Log File.

3 In the Open dialog box, browse to the folder that contains the log file, select a file name, and click Open.

The displayed event list is replaced by the contents of the selected log file.

### Quick Reference

Ribbon

View tab ➤ Palettes panel ▼ Event Viewer ⬠

Event Viewer Menu

Action ➤ Open Log File

# Exporting an Event List to a Text File

Export an Event Viewer event list to a comma-delimited or tab-delimited text file.

When you export an event list to a file, all the events logged during the current session are written to the file. Events that were logged during the session, but that are not displayed in the event list due to filtering, are also written to the file. For more information about using filters to limit the events displayed in the event list, see Controlling the Events Displayed in the Event Viewer (page 1628).

**To export an event list to a text file**

1  Click View tab ➤ Palettes panel ▼ Event Viewer ⌂ .

2  In the Event Viewer (page 2455), on the Action menu, click Export List.

3  In the Save As dialog box, do the following:
   ■ Browse to the folder where you want to save the exported file.

   ■ Enter a file name.

   ■ Specify a file type, which determines the format the text file is saved in.

   ■ Click Save.

## Quick Reference

Ribbon

View tab ➤ Palettes panel ▼ Event Viewer ⌂

Event Viewer Menu

Action ➤ Export List
OR

# Finding a Specific Event in the Event Viewer

Search the event list to find events that meet specified criteria.

**To find a specific event in the Event Viewer**

1  Click View tab ➤ Palettes panel ▼ Event Viewer ⌂ .

2  In the Event Viewer (page 2455), click View menu ➤ Find.

3  In the Find Event (page 2458) dialog box, specify the criteria that match the event you are searching for.

4  Click Find Next.

5  To search for the next event that matches the specified criteria, click Find Next again.

**Quick Reference**

Ribbon

View tab ➤ Palettes panel ⏷ Event Viewer ⟁

Event Viewer Menu

View ➤ Find

Dialog Box

[Find Event](page 2458)

# Copying An Event to the Clipboard

Copy a single event from the Event Viewer event list to the Windows clipboard for use in another application.

**To copy an event to the clipboard**

1  Click View tab ➤ Palettes panel ⏷ Event Viewer ⟁ .

2  In the Event Viewer (page 2455), select the event from the event list that you want to copy to the clipboard.

3  In the Event Viewer, on the Action menu, click Properties.

4  In the Event Properties (page 2456) dialog box, click  Copy To Clipboard.

The information about the event is copied to the Windows clipboard, and can be pasted into another application.

**Quick Reference**

Ribbon

View tab ➤ Palettes panel ⏷ Event Viewer ⟁

Event Viewer Menu

Action ➤ Properties
OR
⚠

Dialog Box

[Event Properties](page 2456)

# Multi-View Blocks

A multi-view block is an AutoCAD Civil object that can have different representations in different view directions. You create a multi-view block from AutoCAD blocks that represent the different views of the custom object that you are creating.

# Understanding Multi-View Blocks

A multi-view block can represent different types of objects in AutoCAD Civil, such as the following:

■ Plantings, such as trees and shrubs

■ Utilities, such as fire hydrants

■ Vehicles

■ Signage

Multi-view blocks can be displayed differently in each view direction and each display representation. For example, the top view of a multi-view block representing a traffic sign shows the top of the sign. The bottom view shows the bottom of the sign, and there are representations for left, right, front, and back views. Additionally, you can define a different group of displays for each display representation. You can have one set of views for plan view and another one for reflected view.

The top and profile view blocks representing a pine tree might look like:



# Predefined Multi-View Blocks

The content folders of AutoCAD Civil contain a large number of predefined multi-view blocks for a number of design purposes. These contain multi-view blocks of building footprints, landscape symbols, annotation symbols, site-related symbols, and other useful elements.

You can access these multi-view blocks from the AutoCAD Civil MV Blocks Tool Palette. You can also use DesignCenter™ to insert the blocks.

Predefined multi-view blocks are located by default in the *\\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\data\Symbols\Mvblocks* folder.

# Creating Multi-View Block Definitions

Each multi-view block in a drawing is based on a multi-view block definition. The multi-view block definition specifies the display representations in which the block can be viewed, and the views available in each representation. From a multi-view block definition, you can create as many actual multi-view block references as you need.

Multi-view blocks typically represent 3D Civil objects. For example, you can create a customized, multi-view fire hydrant representation by drawing plan, elevation, and model views, and save each view as an individual AutoCAD block. You assign each block to a view when you create the multi-view block definition. The blocks

(top, bottom, front, back, left, right, and model views) are used to define the custom object as a single multi-view block. By adding points to your views on the defpoints layer prior to defining the block, you can identify additional insertion points for the view block. You set the display representations and view directions when you define the multi-view block.

You can see the individual blocks of your custom object in plan, elevation, and isometric views as one assembly after inserting them in the drawing as a multi-view block. For more information about working with blocks, see the online *AutoCAD User's Guide*.

## Process Overview: Creating Multi-View Block Definitions

There are four general steps for creating a multi-view block:

1 Create a set of view blocks for each view direction in each display representation that you need.

2 Create a multi-view block definition where you assign the view blocks to view directions in individual display representations.

3 If you want the new multi-view block definition to be available in DesignCenter, add the definition to DesignCenter.

4 If you want the new multi-view block definition to be available from a tool palette, add it to a palette.

## Creating View Blocks

Create AutoCAD blocks and use them to represent each view of the multi-view block definition.



**Creating individual blocks for multi-view block definition**

**To create the AutoCAD blocks used to represent each view of the multi-view block**

1   Draw the views you need for a specific display representation:

   ■   To create views for the front and back directions, draw them on the XZ plane.

   ■   To create views for the left and right directions, draw them on the YZ plane.

   ■   To create views for the top and bottom directions, draw them on the XY plane.

2   Specify additional insertion points on the defpoints layer in the AutoCAD point command, if necessary.

   **NOTE** The points added to view blocks are cumulative. For example, if you add one point to a view block used for the top view and two points to the view block used for the model view, you have a total of four points to cycle through. The fourth point is the regular basepoint defined during the creation of the block.

3   Make sure the current coordinate system is the world coordinate system before making blocks from these individual views.

   For more information about coordinate systems, see "Use Coordinates and Coordinate Systems" in the online *AutoCAD User's Guide*.

4   Define each view as a block, and specify the location of the insertion base point as you define each block.

   For example, if you specify the insertion base point of the model view block as the dead center of the top side, the front, back, left, and right view blocks will all have an insertion base point at the midpoint of the bottom edge. In the following illustration, the insertion points for the top, front, and side view blocks are indicated by an "X". For more information about blocks, see "Create and Use Blocks" in the online *AutoCAD User's Guide*.

**TIP** It is helpful to have a naming convention as you save your views as blocks. For example, name the plan view block hydrant-p, and name the right view block hydrant-r.

## Creating a New Multi-View Block Definition

Create a multi-view block definition from existing view blocks created to represent different views of a custom object.

**To create a multi-view block definition**

1   Click Insert tab ➤ Blo ck panel ⬇ Edit Multi-View Block Definition.

   **NOTE** If there is already a multi-view block in the drawing, press Enter when the prompt Select Multi-View Blocks is displayed.

2   In the New Multi-View Block Definition dialog box, enter a name for the new multi-view block definition. Click OK.

3   In the Multi-View Block Definition Properties dialog box, set up the new multi-view block definition:

   ■   To connect view blocks representing different display representations, see Adding View Blocks to the Multi-View Block Definition (page 1635).

   ■   To add notes and files to the multi-view block definition, see Attaching Hyperlinks, Notes, or Files to a Multi-View Block (page 1642).

4   Click OK.

## Quick Reference

Ribbon

Click Insert tab ➤ Block panel ▼ Edit Multi-View Block Definition

Menu

Insert menu ➤ Multi-View Blocks ➤ Definitions

Command Line

AeccMvBlockDefEdit

Dialog Box

Multi-View Block Definition Properties Dialog Box (page 2459)

## Adding View Blocks to the Multi-View Block Definition

Add view blocks corresponding to individual display representations.

You can specify different settings for plan and profile views, for example.



**To add view blocks**

1 Click Insert tab ➤ Block panel ▼ Edit Multi-View Block Definition.

2 In the Multi-View Block Definition Properties dialog box, select the multi-view block definition from the Multi-View Block Name list.

3 Under Multi-View Block Components, select the display representation to which you want to connect view blocks.

There are two display representations available for multi-view blocks:

■ Select 2D for all Plan views.

■ Select 3D for all other views.

4 Click the add button.

5 Use the Select A Block dialog box to select a view block.

6 To specify view blocks in other display representations, repeat steps 4 - 5.

7 Click OK.

**Quick Reference**

Ribbon

Click Insert tab ➤ Block panel ▼ Edit Multi-View Block Definition

Menu

Insert menu ➤ Multi-View Blocks ➤ Definitions

Command Line

AeccMvBlockDefEdit

Dialog Box

Multi-View Block Definition Properties Dialog Box (page 2459)

## Adding a Multi-View Block Definition to DesignCenter

Add a multi-view block to DesignCenter. When you add a multi-view block definition to DesignCenter, you can create new multi-view blocks by dragging and dropping them from DesignCenter to your drawing. To add a multi-view block definition to DesignCenter, you need to first create a DWG file that contains a multi-view block definition.

**NOTE** A side effect of this process is that the multi-view block, when inserted using DesignCenter, will be embedded in an enclosing block reference and must be exploded before it will be recognized by AutoCAD Civil as a multi-view block object.

**To add a multi-view block to DesignCenter**

1  Click View tab ➤ Palettes panel ➤ Design Center.

2  In DesignCenter, click the Folders tab.

3  Browse to and select the folder where you would like to place the multi-view block.

4  At the command prompt, enter WBLOCK.

5  In the Write Block dialog box, select the Block option.

6  In the Block list, select the name of the multi-view block definition that you would like to add to the folder selected in DesignCenter.

7  Under Destination, browse to the folder specified in DesignCenter. Click Save in the Browse For Drawing File dialog box.

8  Verify the insert units for the multi-view block drawing file.

**NOTE** When using DesignCenter, the insert units will be used to convert the units assigned to the multi-view block to the units specified for the active drawing.

9  Click OK in the Write Block dialog box to write the multi-view block to the drawing file.

10  In DesignCenter, press the F5 key to refresh the contents. The multi-view block is placed in the selected DesignCenter folder.

# Inserting Multi-View Blocks into a Drawing

Insert multi-view blocks into a drawing by dragging and dropping them from the AutoCAD Civil - MVBlocks Tool Palette, or by referencing their definitions using DesignCenter.

After a multi-view block has been inserted into the drawing, additional references can be added by using the Insert tab ➤ Block panel ▼ Add Multi-View Block command.

Predefined multi-view blocks are located in the \\*Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\data\Symbols\Mvblocks* folder by default.

## Inserting a Multi-View Block Using the Tool Palette

Use the Civil Multiview Blocks Tool Palette to insert a multi-view block into a drawing.

**To insert a multi-view block into a drawing using the Tool Palette**

1 Click Home tab ➤ Palettes panel ➤ Tool Palettes Window.

2 Right-click anywhere on the Tool Palette's vertical bar, and click Civil Multiview Blocks.

3 Locate the block to insert by clicking the named tabs.

4 Click the block and drag it into the drawing window.

5 Specify the insertion point and rotation angle.

**NOTE** Multi-view blocks are embedded in an enclosing block reference when they are initially inserted and must be exploded before they can be recognized by AutoCAD Civil as a multi-view block object.

## Inserting a Multi-View Block Using DesignCenter

Use DesignCenter to insert a multi-view block into a drawing.

**NOTE** Multi-view blocks are embedded in an enclosing block reference when they are initially inserted and must be exploded before they can be recognized by AutoCAD Civil as a multi-view block object.

**To insert a multi-view block into a drawing using DesignCenter**

1 Click View tab ➤ Palettes panel ➤ Design Center.

2 In DesignCenter, select the folder containing the DWG file or files (do not select individual blocks within the DWG files).

3 From the DesignCenter preview pane, drag the image representing the multi-view block DWG file into the drawing.

4 Specify the insertion point, if prompted.

5 Specify the scale, if prompted:
   - To specify a scale value, enter a value at the Command prompt.

   - To dynamically set the scale, use the pointing device, and click when you see the appropriate scale.

6 Specify the rotation angle, if prompted:
   - To specify a rotation value, enter a value at the Command prompt.

■ To dynamically set the rotation, use the pointing device, and click when you see the appropriate rotation

**7** To insert additional references to the multi-view block, repeat steps 3 - 6. 3.

**8** Press Enter to finish inserting blocks.

## Inserting Additional Instances of Multi-View Blocks into a Drawing

After one instance of a multi-view block has been inserted into the drawing, additional instances can be added.

The block must already exist in the current drawing to use this command. Insert the original block using the Civil Multiview Blocks Tool Palette.



**To add another instance of a multi-view block into a drawing**

**1** Click Insert tab ➤ Block panel ▼ Add Multi-View Block.

**2** Use the AutoCAD Properties Palette to select the multi-view block to insert. All available blocks are listed in the Definition list on the Design tab.

> **NOTE** If you do not see the block you want to insert, add it to the drawing by dragging it from the Tool Palette or by adding it from the DesignCenter.

**3** Specify the insertion point, rotation, and other settings. You can enter these values in the Properties Palette, or you can use the command line options.

## Modifying Multi-View Blocks

Change the attributes of a multi-view block in a drawing. For example, you can change the rotation or the scaling factor.

There is a multi-view block contextual ribbon in AutoCAD Civil 2010 that you can use to access commonly-used commands. Select the multi-view block in the drawing to display this ribbon tab.

> **NOTE** If you select a multi-view block in a drawing and the multi-view block contextual ribbon is not displayed, use the EXPLODE command to explode the multi-view block. Multi-view blocks are embedded in an enclosing block reference when they are initially inserted and must be exploded before they can be recognized by AutoCAD Civil as a multi-view block object.

## Changing the Rotation of a Multi-View Block

**To change the rotation of a multi-view block**

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Select the multi-view block.

3 On the Properties palette, expand Basic. Expand Location.

4 Enter a value for Rotation.

## Changing the Associated Multi-View Block Definition

Change the multi-view block definition associated with the multi-view block. For information about the relationship between multi-view blocks and multi-view block definitions, see Creating Multi-View Block Definitions (page 1632).

**To change the multi-view block definition associated with the multi-view block**

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Select the multi-view block.

3 On the Properties palette, expand Basic. Expand General.

4 Click Definition. Select a new multi-view block definition for the multi-view block.

## Changing the Scale Factor of a Multi-View Block

**To change the scale factor of a multi-view block**

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Select the multi-view block.

3 On the Properties palette, expand Basic. Expand Scale.

4 Enter new values for the X, Y, and Z scale factors.

## Changing the Offset of a Multi-View Block

Change the offset of a multi-view block from its insertion point.

When you first add a multi-view block to a drawing, the offset value is set to 0. Change the position of the multi-view block by offsetting it from its original insertion point.

You can also change the offset of a multi-view block using the Properties palette.

**To change the offset of a multi-view block from its insertion point**

1 Select the multi-view block.

2 Click Multi-View Block tab ➤ Modify panel ➤ Edit View Block Offsets.

> **NOTE** If you select a multi-view block in a drawing and the multi-view block contextual ribbon is not displayed, use the EXPLODE command to explode the multi-view block.

**3** Select the Location grip.

In Model view, the Location grip has three possible edit modes: Edit Along XY Plane, Edit Along XZ Plane, and Edit Along ZX Plane. Press Ctrl to cycle among modes until you are in the desired mode. The default is Edit mode along the XY plane. In Plan view, the Location grip is restricted to movement in one plane.

**4** Offset the multi-view block to the desired location and click once, or enter a value and press Enter.

If you want to enter a specific value for the second direction in any edit mode (for example, in the Y direction when editing along the XY plane), press Tab to cycle to the second direction.

You can also lock the movement of the multi-view block along a specific direction. If you enter a value for either of the dimension directions in the current edit mode and then press Tab, the movement of the multi-view block is constrained to the second dimension direction. When editing along the XY plane, for example, you can enter a value for the X dimension, and then press Tab. The X dimension is locked at that value, and movement of the multi-view block is constrained to the Y dimension direction.

**5** Right-click and click Exit Edit View Block Offsets.

The selected view block is offset from the original insertion point.

## Changing the Location of a Multi-View Block

Relocate an existing multi-view block by changing the coordinate values of its insertion point. In Model view, the Location grip has three possible edit modes: Edit Along XY Plane, Edit Along XZ Plane, and Edit Along ZX Plane. Press Ctrl to cycle among modes until you are in the desired mode. The default is Edit mode along the XY plane. In Plan view, the Location grip is restricted to movement in one plane

You can also change the location of a multi-view block using the Properties palette.

**NOTE** This procedure can only be done in Model view.

**To relocate an existing multi-view block by changing the coordinate values of its insertion point**

**1** Select the multi-view block you want to relocate.

**2** Select the Location grip.

Location
Press Ctrl to cycle between:
 - Edit Along Object's XY Plane
 - Edit Along Object's YZ Plane
 - Edit Along Object's XZ Plane

**3** Move the multi-view block to the desired location and click once, or enter a value and press Enter.

If you want to enter a specific value for the second direction in any edit mode (for example, in the Y direction when editing along the XY plane), press Tab to cycle to the second direction.

You can also lock the movement of the multi-view block along a specific direction. If you enter a value for either of the dimension directions in the current edit mode and then press Tab, the movement of the multi-view block is constrained to the second dimension direction. When editing along the XY plane, for example, you can enter a value for the X dimension, and then press Tab. The X dimension is locked at that value, and movement of the multi-view block is constrained to the Y dimension direction.



Polar: < 45.00, Endpoint: < 60.00
Edit Along Object's XY Plane

**4** Right-click and click Deselect All to turn off grips.

## Attaching Hyperlinks, Notes, or Files to a Multi-View Block

Attach hyperlinks, enter notes, or attach reference files to a multi-view block in a drawing. You can also edit notes and edit or detach reference files from a multi-view block.

**To attach hyperlinks, enter notes, or attach reference files to a multi-view block in a drawing**

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Select the multi-view block.

3 On the Properties palette, click the Extended Data tab.

4 To add a hyperlink, click 🔲 next to Hyperlink, and specify the link.

5 To add or edit a note, click 🔲 next to Notes. Enter text. Click OK.

6 Click 🔲 next to Reference documents. Attach, detach, or edit a reference file:

■ To attach a reference file, click 🔲 . Select a file. Click Open.

■ To edit the description of a reference file, enter text under Description.

■ To edit a reference file, double-click the reference file name to start its application.

■ To detach a reference file, select the file name. Click ✖ .

## Editing the Attributes of a Multi-View Block

**To display the attributes defined for a multi-view block**

1 Click View tab ➤ Palettes panel ➤ Properties.

2 Select the multi-view block.

3 On the Properties palette, click the Design tab. Expand Advanced.

4 Click Attributes.

5 Edit attributes as needed.

6 Click OK.

**NOTE** You can also select the multi-view block to display the Multi-View Block ribbon tab, and then use the Edit Attributes command on the Modify panel to edit the attributes.

## Updating Multi-View Block Attributes

Use this procedure to update the attributes of blocks in your drawing, using the current attributes defined for the view blocks upon which each multi-view block is based. You need to do this if you change attribute definitions in one or more of the block definitions on which the multi-view block is based.

**To update attribute definitions**

1 Select the multi-view block.

**2** Click Multi-View Block tab ➤ Modify panel ➤ Update 🖼️.

**3** Enter **y** (Yes) to include text style properties (for example, width factor, oblique, mirroring) when updating attributes, or **n** (No) to update all attributes properties except those relating to text style.

**4** Press Enter to end the update process.

# Drafting Tools

You can perform a variety of editing operations on linework components and certain Civil objects using the AEC Modify Tools.

This suite of tools can be used on any of the following objects and is available on the shortcut menu regardless of whether any of these objects of these types are selected:

■ Hatches

■ Polylines

■ Circles

■ Arcs

■ Text/Multiline text (Mtext)

■ Any of the above when embedded in blocks

Although the modify tools can only be applied to the types of objects listed, objects of any type can be used as reference points or boundaries for the editing operations. For example, linework can be trimmed or extended in relation to Civil objects. Likewise, Civil objects can be used to define a cropping boundary to subtract from linework. For more information about the applicability of the various tools, refer to the individual procedures.

# Trimming Linework Objects

You can trim hatches, lines, polylines, arcs, circles, or any block-based content created from these types of linework or objects.

**To trim linework**

**1** Select the linework, object, or block you want to trim.

**2** Right-click and click AEC Modify Tools ➤ Trim.

**3** Specify two points to establish the trim line.

The trim line changes color after the second point is selected, as does all linework (except hatches) on the side of the trim line where the cursor is positioned.

**4** Click the side of the trim line where the region you want to remove is located.

All linework on the selected side of the trim line is removed. Boundaries for any closed polylines on the other side of the trim line are redrawn along the trim line.

existing block

select block

specify first trim
line point

specify second trim
line point

specify side to trim

result

**Trimming a block**

# Trimming Linework Objects to an Edge

Trim one or more lines, polylines, arcs, circles, or hatches precisely to a selected edge or side of any straight segment of linework.

**TIP** If you need to trim linework to the edge of an arced segment of an object, use either the Crop option (see Cropping Linework Objects (page 1650)) or the Subtract option (see Subtracting from Linework Objects (page 1647)). The Crop option removes linework outside the implied boundary of the arc, while the Subtract option removes linework inside the implied boundary of the arc.

**To trim one or more objects to an edge**

1  Select the linework or object you want to trim.

2  Right-click and click AEC Modify Tools ➤ Trim.

3  Press Enter.

4  Move the cursor over the edge or line to which you want to trim the selected linework or object.

5  Click when the trim line is displayed.

6  Click the side of the trim line you want trimmed.
   All selected linework is trimmed to the edge you selected.



existing wall
and polyline

select polyline

specify trim edge

specify side
to trim

result

**Trimming linework to the
edge of a wall**

# Extending Linework to an Edge

Extend one or more lines, open polylines, or arcs precisely to an implied edge of a straight or arced segment
of any object, or to an implied extension of that segment.

**To extend linework to an edge**

1 Select the linework you want to extend.

2 Right-click and click AEC Modify Tools ➤ Extend.

3 Move the cursor over the edge or line to which you want to extend the selected linework. Click when a temporary line is displayed extending along and beyond the edge/line in both directions.

4 Click any point on the linework you want to extend or click the edge or line to which you want to extend the linework.

The linework is extended to the edge. If you selected multiple lines, clicking on any one of them extends them all.



existing wall
and polyline

select polyline

specify trim edge

select point near
end to extend

result

**Extending linework to the
edge of a wall**

# Dividing Linework Objects

Divide linework or a block containing linework into two (or more) parts defined by a dividing line you draw. You can then select the parts on either side of the line. If a closed-boundary object is divided into two parts using this option, both parts are recreated as closed. If an open polyline in a zigzag pattern is divided along its length, many selectable segments are created.

**To divide linework**

1 Select the linework, object, or block you want to divide.

2 Right-click and click AEC Modify Tools ➤ Divide.

3 Specify two points to establish the dividing line. In the case of a closed-boundary object, the points can be inside or outside the boundaries. The dividing line is automatically extended until it intersects all boundaries of the selected object.

You can select the divided parts of the linework or object on either side of the divide line. The divide line is not visible after you select the second point.

existing space

select space

specify divide line
start point

specify divide line
endpoint

result

**Dividing a space object**

# Subtracting from Linework Objects

Subtract a selected part from linework or from a block containing linework without having to redraw any of the remaining components. The part to be subtracted can contain or be delineated by lines, polylines

(closed or open), arcs, circles, text, or hatches. If the subtraction removes part of a closed polyline, the polyline is recreated as closed. If the subtraction bisects a closed polyline, both parts are recreated as separate closed polylines. However, when the subtraction bisects a hatch, the hatch remains a single linework entity that you can select by clicking either part.

---

**NOTE** Anything subtracted by this operation is actually deleted from the drawing. If you want to hide linework from view, you may want to use the Obscure command instead. For more information, see Obscuring Regions of Linework (page 1648).

---

**To subtract a selected part from an object**

1   If it does not already exist, draw the linework that delineates what you want to subtract.

2   Select the linework or object from which to subtract the delineated part.

3   Right-click and click AEC Modify Tools ➤ Subtract.

4   Select the linework delineating the part you want to subtract. Press Enter.

5   Press Enter to retain the selected linework. Enter y (Yes) to erase it.



existing block
and polyline

select block

select object
to subtract

result with
linework retained

result with
linework erased

**Subtracting part of a block**

# Obscuring Regions of Linework

Obscure or "hide" a region of linework in relation to other linework. Use this feature when you want to indicate that one entity extends behind or beneath another entity that is displayed in the foreground. The obscured linework is placed on the layer specified by the Hidden layer key.

**To obscure a region of linework in relation to other linework**

1 Select the linework you want to obscure.

2 Right-click and click AEC Modify Tools ➤ Obscure.

3 Select the foreground linework that will obscure the linework selected in step 1.

4 Press Enter.

The obscured region is delineated with a dashed line. You can select it (or its component segments) separately from the linework it was part of before being obscured.



existing blocks

select linework
to hide

select obscuring
linework

result

**Obscuring linework with a block**

# Merging Linework Objects

Merge contiguous closed polylines, circles, hatches, or blocks into a single closed-boundary object or linework entity.

**To merge linework**

1 Select the linework or object to which you want to add one or more other linework entities or objects.

2 Right-click and click AEC Modify Tools ➤ Merge.

3 Select the linework/object(s) you want to merge with the originally selected linework/object, and press Enter.

4 Press Enter to retain the selected linework. Enter y (Yes) to erase it.

All selected linework and/or objects are merged into a single entity.

existing block
and polyline

select block

select object
to merge

result with
linework retained

result with
linework erased

**Merging a region with a block**

# Cropping Linework Objects

Crop linework to a smaller area using a boundary you define. You can crop hatches, lines, polylines, arcs, circles, or blocks. Neither the linework to be cropped nor the linework that defines the new boundary needs to have closed boundaries. For example, you can either use a v-shaped polyline to crop a circle or a circle to crop a v-shaped polyline.

**To crop linework using a boundary**

1  Draw the linework that defines the new boundary to which you want to crop the existing linework.

2  Right-click and click AEC Modify Tools ➤ Crop.

3  Select the linework to crop and press Enter.

4  Select the linework you want to use to define the new boundary (the linework you drew in Step 1), and press Enter.

5  Press Enter to retain the selected linework (the crop boundary), or enter y (Yes) to erase it.

   All of the originally selected linework outside the crop boundary is erased. If you choose to erase the crop boundary, boundaries are redrawn for any closed-boundary entities inside the crop area. But any open-boundary objects that have been cropped remain open.

existing block
and polyline

select block

select crop
boundary

result with
linework retained

result with
linework erased

# Utilities Command Reference

You can use commands to quickly access AutoCAD Civil utility functionality.

The following table lists the utility-related AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| AeccMvBlockDefEdit | Edits a multi-view block definition (page 1632) |
| AecMvBlockEditAttributes | Edits the attributes defined for a multi-view block (page 1642) |
| AecMvBlockEditViewBlockOffsets | Changes the offset of a multi-view block from its insertion point (page 1639) |
| CgADist | Calculates the total of several disjunct distances (page 1604) |
| CgAng | Displays the acute and obtuse angle between points or intersecting lines (page 1606) |
| CgCDist | Displays the distance between points (page 1605) |
| CgList | Displays coordinate geometry for lines and arcs (page 1606) |
| CgSList | Displays the slope, grade, and horizontal distance between two points (page 1607) |

| Command | Description |
|---|---|
| ContentBrowser | *Opens the Content Browser, which is a library of tool catalogs, including subassembly catalogs* |
| DockCT | Docks the Coordinate Tracker utility (page 1623) |
| DockInquiry | Docks the Inquiry utility (page 1618) |
| FloatCT | Changes the Coordinate Tracker utility from docked to floating (page 1623) |
| FloatInquiry | Floats the Inquiry utility (page 1618) |
| HideCT | Hides the Coordinate Tracker utility (page 1623) |
| HideEventViewer | Hides the Event Viewer utility (page 1626) |
| HideInquiry | Hides the Inquiry utility (page 1618) |
| Mapcheck | Displays the Mapcheck Analysis utility (page 1607) |
| MvBlock Update | Updates the attributes of blocks in your drawing (page 1642) |
| MvBlockAdd | Inserts a multi-view block definition into a drawing (page 1637) |
| Notes | Adds notes to a drawing (page 1616) |
| ObjectViewer | Displays the Object Viewer utility (page 1617) |
| ShowCT | Displays the Coordinate Tracker utility (page 1623) |
| ShowEventViewer | Displays the Event Viewer utility (page 1626) |
| ShowInquiry | Displays the Inquiry utility (page 1618) |

# File and Data Sharing

# Sharing Drawings and Data With Other Applications

# 39

AutoCAD Civil provides many utilities for sharing your drawings and data with other applications.

## Comparing Sharing Methods

To share an AutoCAD Civil drawing data with people who do not have AutoCAD Civil, you can use proxy graphics, the Object Enabler, the Export To AutoCAD commands.

The following table lists the features of each method:

| Data Sharing Method | Use of objects in other AutoCAD-based applications | Status of objects in other applications | 2D and 3D style display settings honored in different viewports | Objects still exist when reopened in AutoCAD Civil |
|---|---|---|---|---|
| **Proxy Graphics** | View/plot objects and design data; no manipulation of objects. | Objects are displayed as proxy graphics with no intelligence. | No | Yes |
| **Object Enabler** | View, plot, copy, and manipulate object labels. Design data is not editable. | Objects are maintained as intelligent objects. | Yes | Yes |
| **Export To AutoCAD (DWG or DXF)** | Objects are exploded to AutoCAD primitives. The visual integrity of the model is preserved. Object design data is not available. | No objects in other applications. Only AutoCAD primitives. | No | No |

| Data Sharing Method | Use of objects in other AutoCAD-based applications | Status of objects in other applications | 2D and 3D style display settings honored in different viewports | Objects still exist when reopened in AutoCAD Civil |
|---|---|---|---|---|
| Export to SDF | Points, parcels, alignments, pipes and structures are exported to the SDF file. Can be used in applications that can read SDF. AutoCAD Map can import this data. | Primarily a format for AutoCAD Map. | SDF is a separate file. Original drawing is not affected. | SDF is a separate file. Original drawing is not affected. |
| Export to 3D DWF | Uses the 3D display representations of the objects for display. Some objects do not have a 3D display component so are not displayed in 3D DWF. There is also object meta-data exported with 3D DWF. All of the objects export the meta-data. | Autodesk Design Review or DWF Viewer is required. | Autodesk Design Review or DWF Viewer is required. 3D DWF uses the 3D display component when exported. | Original drawing is not affected. |

# Using Proxy Graphics

Proxy graphics store the last viewed image of AutoCAD Civil objects, which lets users view your drawing without modifying the original objects. Proxy graphics store only the display representation for the current active viewport.

**NOTE** Depending on the complexity of the drawing, a drawing saved with proxy graphics can become extremely large, slowing drawing performance. It is recommended that you make a copy of the drawing without proxy graphics before turning them on in the drawing you want to share with other users.

**To generate proxy graphics**

1 Set up the drawing to display the view for which you want to create proxy graphics.

2 At the command line, enter **Proxygraphics**.

3 Enter **1**.

4 Save the drawing.

5 To turn off proxy graphics, enter **Proxygraphics**, and then enter **0**.

Command Line

Proxygraphics

# Using the Object Enabler

Users who do not have AutoCAD Civil but who have an AutoCAD 2010-based product can install the Object Enabler and view your drawing exactly as you created it.

The Object Enabler is available on the AutoCAD Civil installation CD and on the Autodesk web site at *www.autodesk.com/aecobjenabler*.

After a user installs the Object Enabler, the user can open the drawing and view, plot, manipulate, and copy AutoCAD Civil objects using standard AutoCAD commands. AutoCAD Civil objects are maintained as intelligent objects, and the drawing can be re-used in AutoCAD Civil.

# Exporting a Drawing to AutoCAD

You can export a drawing created in AutoCAD Civil to AutoCAD®. This process creates either a new AutoCAD 2010, 2007, 2004, 2000 or R14 format drawing with all civil objects exploded to AutoCAD primitives. The original drawing is not affected.

When you open the resulting drawing, the primitive objects are not restored to their original state.

The following format compatibility rules apply.

| Format ... | Compatible with AutoCAD ... |
|---|---|
| 2010 | 2010 |
| 2007 | 2007-2009 |
| 2004 | 2004-2006 |
| 2000 | 2000-2003 |
| R14 | R14 |

The Export To AutoCAD commands respect the current display settings of the objects in your drawing. For example, if a surface in the drawing is set up to display the border and contours only, the drawing created with the ExportTo Auto CAD command will contain primitives that represent the surface border and contours only.

If the current drawing contains multiple viewports, the Export To AutoCAD commands create the primitives based on the view of the objects in the active viewport.

---

**TIP** For information about the Save As AutoCAD DWG command and how it is different from the Export To AutoCAD commands, see *Save A Drawing* and *Save As* in the AutoCAD Help.

---

**To export a drawing to AutoCAD**

1  Verify that the drawing's Model tab is selected.

2  At the command line, enter **UCS**, and then enter **World**.

**3** Click  ➤ Export ➤ AutoCAD DWG, and then click the version you want to use.

**4** Enter a filename, and specify a path, if necessary.

The default filename includes any prefix or suffix that you specified on the AEC Editor tab of the Options dialog box. For more information, see Changing the AEC Editor Settings (page 1658).

To change export options, such as how the command handles xrefs, use the command line version of this command. For more information, see Exporting a Drawing to AutoCAD (page 1657).

**5** Click Save.

---

**TIP** You can use the Autodesk Civil Batch Drawing Converter Utility to convert multiple drawings to a legacy AutoCAD drawing format. See Using the Civil Batch Drawing Converter (page 1287).

---

When converting AutoCAD Civil drawings to legacy AutoCAD drawing formats, the application does not convert AutoCAD Civil objects to the previous version.

### Quick Reference

Application Menu

 ➤ Export ➤ AutoCAD DWG

Command Line

ExportToAutoCAD2010
ExportToAutoCAD2007
ExportToAutoCAD2004
ExportToAutoCAD2000
ExportToAutoCADR14

## Exploding AutoCAD Civil Objects

All AutoCAD Civil objects support the AutoCAD Explode command. In addition, labels on objects can be individually exploded.

For more information on the Explode command, see *Explode*.

A setting in the AutoCAD Options dialog box affects how AutoCAD Civil objects are exploded. For more information, see Changing the AEC Editor Settings (page 1658).

## Changing the AEC Editor Settings

You can specify settings for the Export To AutoCAD commands, including how the commands handle XRefs.

**To change the AEC Editor Settings**

**1** At the command line, enter **Options**.

**2** In the AutoCAD Options dialog box, click the AEC Editor tab (page 2453).

**3** Under Block Properties Of Exploded Objects, select Maintain Resolved Layer, Color, Linetype to maintain the layer, color, and linetype of objects when you explode them.

**4** Under Export To AutoCAD, select Bind Xrefs if you want to bind the xrefs to the host drawing when exploding, thus creating local block definitions. If you select this option, all xrefs and their dependent named objects become a part of the exported drawing.

**5** If you select the Bind Xrefs check box, then either select or clear the Use Insert Method When Binding Xrefs check box.

**6** Enter a prefix or a suffix to be added to the drawing filename when the drawing is exported with the Export To AutoCAD commands.

---

**NOTE** The other options on the AEC Editor tab are not supported by AutoCAD Civil.

---

**7** Click OK.

### Quick Reference

Application Menu

 ➤ Options ➤ AEC Editor tab

Command Line

Options

Dialog Box

AEC Editor Tab (Options dialog box) (page 2453)

# Exporting a Drawing to a DXF File

You can export a drawing created in AutoCAD Civil to AutoCAD. This process creates either a new AutoCAD 2010, 2007, 2004, 2000, or R12 DXF format drawing with all civil objects exploded to AutoCAD primitives. The original drawing is not affected.

A DXF file is a text file that contains drawing information that other CAD systems can read. When you open the DXF file, the exploded objects are not restored to their original state.

You can export drawings to the following DXF formats:

■ AutoCAD 2010 DXF

■ AutoCAD 2007 DXF

■ AutoCAD 2004 DXF

■ AutoCAD 2000 DXF (including 2000i and 2002)

■ AutoCAD Release 12 DXF

This process creates a new DXF file with all AutoCAD Civil objects exploded to AutoCAD primitives. The original drawing is not affected.

---

**NOTE** When you use the Export To AutoCAD commands, all AutoCAD Civil objects are exploded to AutoCAD primitives. Opening the drawing in AutoCAD Civil does not restore these primitive objects to AutoCAD Civil objects.

---

**To export a drawing to a DXF file**

**1** Verify that the drawing's Model tab is selected.

**2** At the command line, enter **UCS**, and then enter **World**.

**3** Click  ➤ Export ➤ AutoCAD DXF.

**4** Enter a filename, and specify a path, if necessary.

The *.dxf* extension for the file name is specified by default. The default file name also includes any prefix or suffix that you specified on the AEC Editor tab of the Options dialog box. For more information, see Changing the AEC Editor Settings (page 1658).

**5** Click Save.

### Quick Reference

Application Menu

 ➤ Export ➤ AutoCAD DXF

Command Line

ExportToAutoCAD2010Dxf
ExportToAutoCAD2007Dxf
ExportToAutoCAD2004Dxf
ExportToAutoCAD2000Dxf
ExportToAutoCADR12Dxf

# Exporting Civil Data to SDF Files

Use the Export to SDF command to export AutoCAD Civil data to Spatial Data Format (SDF) files.

Spatial data format files can be used in AutoCAD Map 3D and Autodesk MapGuide Enterprise. The alignment, point, parcel, and pipe network data are converted to data objects with attributes.

The AutoCAD Civil alignment, point, parcel, and pipe network data is converted to data objects with attributes. SDF files store data as features, which means that you are dealing with real-world objects rather than geometry.

| AutoCAD Civil Object | Exported Attributes |
|---|---|
| Alignment | ■ Alignment Name |
| | ■ Length |
| | ■ Design Speed |
| | ■ Starting Station |
| | ■ Ending Station |
| Point | ■ Point Number |
| | ■ Point Name |
| | ■ Elevation |
| | ■ Description |
| | ■ Raw Description |
| | ■ Latitude |

■ Longitude

| Parcel | ■ Parcel Name |
| | ■ Parcel Number |
| | ■ Area |
| | ■ Perimeter |

| Pipe | ■ Network Name |
| | ■ Length |
| | ■ Slope |
| | ■ Inside Diameter |
| | ■ Outside Diameter |
| | ■ Shape |
| | ■ Elevation at Start |
| | ■ Elevation at End |
| | ■ Structure Start |
| | ■ Structure End |

| Structure | ■ Network Name |
| | ■ Rim Elevation |
| | ■ Structure Name |

**To export AutoCAD Civil drawing data to an SDF file**

1  Click Output tab ➤ Export panel ➤ Export Civil Objects To SDF  .

2  In the Export To SDF (page 2463) dialog box, specify a name for the SDF file.

3  Specify the coordinate system.

> **NOTE** If the drawing already has a coordinate system specified, it is used automatically and the Select Coordinate System controls in the Export To SDF dialog box are grayed out.

4  Click OK to export the file.

## Quick Reference

Ribbon

Output tab ➤ Export panel ➤ Export Civil Objects To SDF

Command Line

ExportToSDF

Dialog Box

Export To SDF (page 2463)

# Exporting a Drawing to a 3D DWF File

You can export a drawing created in AutoCAD Civil to 3D DWF format.

A Design Web Format (DWF) file is a set of drawings or images that is compressed into a single, smaller file, making sharing across the Web faster and more secure.

The Export to 3D DWF command creates a DWF file with all or the selected AutoCAD Civil objects and AutoCAD primitives. The command uses the 3D display representations of the objects when displaying them in the DWF file. Some objects do not have 3D display components (for example, profile views), so they are not displayed in the 3D DWF.

Object metadata is also exported with 3D DWF. For example, the maximum surface elevation information is available when you view the 3D DWF in the DWF Viewer or Autodesk Design Review.

**NOTE** To export to 2D DWF format, use the Plot feature. For more information see *Create and Modify a Drawing Set for Publishing*.

For more information on 3D DWF publishing, see *3DDWFPublish*.

**To export a drawing to a DWF file**

1  Verify that the drawing's Model tab is selected.

2  At the command line, enter **UCS**, and then enter **World**.

3  Click ➤ Export ➤ Other Formats.

4  In the Files of Type drop-down, click 3D DWF, enter a filename, and specify a path, if necessary.

   The *.dwf* extension for the file name is specified by default.

5  Optionally, click Tools ➤ Options and modify the publishing options. For more information, see DWF Publishing Options Dialog Box (page 2464).

6  Click OK to close the DWF Publishing Options dialog box and click Save to generate the DWF file.

## Quick Reference

Application Menu

➤ Export ➤ Other Formats

Command Line

 3DDWFPublish

# Other Data Sharing Methods

Share AutoCAD Civil data using additional utilities and commands.

The following section list additional methods for sharing AutoCAD Civil data.

### Exporting to Google Earth

Use the Export to Google Earth utility to publish and view your design data in Google Earth. For more information, see Publishing Civil Data to Google Earth (page 1669).

### Exporting to DEM

Use the Export Surface to DEM utility to export surface data to a DEM file. The file can be of type USGS (*.dem*) or GeoTIFF (*.tiff*). For more information, see Exporting to DEM (page 714).

### Exporting to LandXML

Use the LandXML export commands to export drawing data to LandXML format. For more information, see Understanding LandXML Import and Export (page 1673).

### Exporting Points

Use the Points export command to export drawing points to an ASCII (text) file or a Microsoft® Access *.mdb* file. For more information, see Exporting Point Data (page 505).

# Drawing Sharing Command Reference

You can use commands to quickly access file and drawing sharing drawing functionality.

| Command | Description |
| --- | --- |
| 3DDWFPublish | Exports an AutoCAD Civil drawing to 3D DWF format (page 1662) |
| ExportToAutoCAD2000 | Exports an AutoCAD Civil drawing to AutoCAD 2000 format (page 1657) |
| ExportToAutoCAD2000Dxf | Exports an AutoCAD Civil drawing to AutoCAD 2000 DXF (page 1659) |
| ExportToAutoCAD2004 | Exports an AutoCAD Civil drawing to AutoCAD 2004 format (page 1657) |
| ExportToAutoCAD2004Dxf | Exports an AutoCAD Civil drawing to AutoCAD 2004 DXF format (page 1659) |
| ExportToAutoCAD2007 | Exports an AutoCAD Civil drawing to AutoCAD 2007 format (page 1657) |
| ExportToAutoCAD2007Dxf | Exports an AutoCAD Civil drawing to AutoCAD 2007 DXF format (page 1659) |
| ExportToAutoCAD2010 | Exports an AutoCAD Civil drawing to AutoCAD 2010 format (page 1657) |
| ExportToAutoCAD2010Dxf | Exports an AutoCAD Civil drawing to AutoCAD 2010 DXF format (page 1659) |
| ExportToAutoCADR12Dxf | Exports an AutoCAD Civil drawing to AutoCAD R12 format (page 1659) |
| ExportToAutoCADR14 | Exports an AutoCAD Civil drawing to AutoCAD R14 format (page 1657) |
| ExportToSDF | Exports an AutoCAD Civil drawing to SDF format (page 1660) |

# Google Earth Import and Export

# 40

Use AutoCAD Civil to retrieve image and terrain data from Google Earth as well as publish and view your design data in Google Earth.

## About Using Google Earth

Use AutoCAD Civil in conjunction with Google Earth to import image and terrain data and publish and view design data.

■ Import surface images and terrain data from Google Earth to better visualize the future development environment. See Importing a Google Earth Image to AutoCAD Civil (page 1665), Importing Google Earth Terrain Data into a Surface (page 1666), and Importing Google Earth Image and Terrain Data (page 1667).

■ Publish the model space entities from AutoCAD Civil to Google Earth to quickly view your design data in the Earth terrain imagery. See Publishing Civil Data to Google Earth (page 1669).

■ Import the elevation data for the current Google Earth view as a mesh. See Importing Google Earth Elevation Data as a Mesh (page 1668).

■ Attach time-related information to each piece of the model data and then display published model data based on start and end times. See Attaching Time Information to Model Data (page 1670).

**NOTE** You can use the imported Google Earth data for demonstration purposes only. Images imported from Google Earth are low resolution and are not suitable for a land survey. In the same manner, publishing Civil data to Google Earth can only serve as a conceptual planning and design tool.

## Importing a Google Earth Image to AutoCAD Civil

You can import Google Earth images into your AutoCAD Civil drawing.

You can manually specify the image insertion point or you can insert the image using the defined coordinate system in the drawing. Before you start, ensure that no tilting, turning, or rotation have been applied to the image.

**To import a Google Earth image into a Civil project**

1 In Google Earth, fly to a point of interest (see the Google Earth User Guide for more information).

The image that is displayed in the Google Earth 3D Viewer is the source image that will be imported. See the *Google Earth User Guide* for more information.

2 Click Insert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image  .

3 Follow the instructions at the command prompt and, depending on whether your drawing has a coordinate system defined (see Specifying Units and Zone Settings (page 63)), do one of the following:

■ Choose to locate the image using the defined coordinate system or manually identify the image location.

■ Manually specify the image insertion point and define image rotation angle by entering the corresponding values or clicking in the drawing workspace.

**NOTE** If a map projection coordinate system is defined, AutoCAD Civil can automatically place the image or terrain data in the proper location in the drawing.

The image is displayed in the drawing as a new object. The image file name is generated and the image is saved in JPEG format in the same folder as the current drawing file. You can use the imported image for adding design objects and data.

**NOTE** The imported image is displayed as grayscale in the application.

### Quick Reference

Ribbon

Insert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image 

Menu

File ➤ Import ➤ Import Google Earth Image

Command Line

ImportGEImage

## Importing Google Earth Terrain Data into a Surface

Create a AutoCAD Civil surface that represents a specific location on the surface of the Earth by importing terrain data from a Google Earth image.

You can insert the surface using the defined coordinate system or by selecting the previously imported image. Before you start, ensure that no tilting, turning, or rotation have been applied to the image.

**To create a surface using the image terrain data from Google Earth**

1 In Google Earth, fly to a point of interest (see the *Google Earth User Guide* for more information). The image that is displayed in the Google Earth 3D Viewer is the source image whose terrain data will be imported.

2 Click Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Google Earth  .

3 Follow instructions in the command prompt and, depending on whether your drawing has a coordinate system defined (see Specifying Units and Zone Settings (page 63)), do one of the following:

■ Choose to locate the surface using the defined coordinate system or by selecting the previously imported image.

■ Locate the surface using the previously imported image.

---

**NOTE** If a map projection coordinate system is defined, AutoCAD Civil can automatically place the image or terrain data in the proper location in the drawing.

---

4 In the Create Surface dialog box, follow the steps for creating a TIN surface. See Creating a TIN Surface (page 601).

A surface is created. It is displayed in AutoCAD Civil drawing and the Prospector tab. The location of the surface points is based on the closest values obtained from the transformation of the Google Earth geographical coordinates to the linear coordinates of the drawing.

### Quick Reference

Ribbon

Home tab ➤ Create Ground Data panel ➤ Surfaces drop-down ➤ Create Surface From Google Earth 

Menu

File ➤ Import ➤ Import Google Earth Surface

Command Line

ImportGESurface

## Importing Google Earth Image and Terrain Data

You can import Google Earth images and create a AutoCAD Civil surface based on the terrain data from the image location in one step. The location of the surface points is based on the closest values obtained from the transformation of the Google Earth geographical coordinates to the linear coordinates of the drawing.

The image that is displayed in the Google Earth 3D Viewer is the source image whose terrain data will be imported. Before you start, ensure that no tilting, turning, or rotation have been applied to the image.

**To create a surface and import an image using the Google Earth data**

1 In Google Earth, fly to a point of interest (see the *Google Earth User Guide* for more information).

2 Click Insert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image And Surface  .

3 Follow the instructions at the command prompt and, depending on whether your drawing has a coordinate system defined (see Specifying Units and Zone Settings (page 63)), do one of the following:

■ Choose to locate the image and surface using the defined coordinate system or manually identify the image location.

■ Manually identify the image and surface location and orientation.

**Quick Reference**

Ribbon

Insert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image And Surface 

Menu

File ➤ Import ➤ Import Google Earth Image And Surface

Command Line

ImportGEData

# Importing Google Earth Elevation Data as a Mesh

You can import the elevation data for the current Google Earth view as a polygon mesh.

The image that is displayed in the Google Earth 3D Viewer is the source image whose elevation data is imported as a polygon mesh.

Before you start, ensure that no tilting, turning, or rotation have been applied to the source image. For best results, zoom in the displayed view, so that surface terrain details are visible.

**To import a Google Earth mesh**

---

**IMPORTANT** Before importing Google Earth elevation data, maximize the Google Earth application window.

---

1 In Google Earth, fly to a point of interest (see the *Google Earth User Guide* for more information).

2 In AutoCAD Civil, clickInsert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image And Surface  .

3 In the drawing, specify the insertion point and rotation angle to orient the image. The mesh is imported into the drawing.

---

**NOTE** If a map projection coordinate system is defined, AutoCAD Civil can automatically place the image or terrain data in the proper location in the drawing.

---

AutoCAD Civil creates a render material, using the displayed Google Earth 3D Viewer image. This render material is associates with the current drawing and assigned to the created mesh.

The image captured from the Google Earth view is saved in the same folder as the current drawing. The image filename inherits the first three letters from the drawing file name.

4 Optionally, to view the image draped on the mesh, select Realistic from the Type list in the Material Editor of the Materials window. For more information, see Object Rendering (page 1577) and Draping Images On Surfaces (page 695).

When importing the Google Earth elevation data to a AutoCAD Civil drawing, note the following:

■ When you place the mesh into the drawing, you can use the AutoCAD MOVE and ROTATE commands to refine the position of the mesh relative to your model. Do not adjust the position of your model. Rather, adjust the position of the mesh relative to your model.

■ The dimensions of the mesh are defined by the extent of longitude and latitude that the mesh covers in Google Earth, and transformation of these extents to the linear units of the drawing.

■ The mesh size is fixed at 32 rows by 32 columns.

■ The material created in the drawing is named using the first three letters of the drawing file, followed by several random numbers.

**Quick Reference**

Ribbon

Insert tab ➤ Import panel ➤ Google Earth drop-down ➤ Google Earth Image And Surface 🗺️

Menu

File ➤ Import ➤ Import Google Earth Mesh

Command Line

ImportGEMesh

# Publishing Civil Data to Google Earth

Use the Google Earth publishing wizard to export the 3D design data from AutoCAD Civil to Google Earth for quick visualization of the model within the context of the Earth surface imagery.

After you prepare your AutoCAD Civil drawing data, run the Publish to Google Earth command and follow the steps in the publishing wizard.

**To publish 3D design data to Google Earth**

1   Click Output tab ➤ Publish panel ➤ Publish To Google Earth 🌐 .

2   On the Describe page, specify information about the published drawing. See the Describe page of the wizard (page 1869).

3   On the Items page, specify the model space entities that you want to publish. See the Items page of the wizard (page 1869).

   **NOTE** When there are render materials attached to model space entities in AutoCAD Civil, you can publish the materials with the entities. If timespan information is attached to model space entities, this information is not published to Google Earth when you select to publish render materials. See Attaching Time Information to Model Data (page 1670).

4   On the Geo-Reference page, use the controls to transform the linear coordinates of the drawing to geographic coordinates used in Google Earth. See the Geo-Reference page of the wizard (page 1870).

5   On the Nudge page, fine-tune the location of the created model. See the Nudge page of the wizard (page 1871).

6   On the File page, specify the name and location for your published file. See the File page of the wizard (page 1872).

7   Click Publish and view the status of the publishing operation.

8   After the operation is finished, click View to view the published model in Google Earth.

   **NOTE** If Google Earth is not running, it is launched and the published model is displayed in the Google Earth 3D Viewer.

The resulting .kml or .kmz file is stored in the location that you specified. You can share the published file with remote project participants, who can view it using Google Earth.

**NOTE** You can control the appearance of the published data using the Google Earth controls. For more information, see the Google Earth User Guide.

### Quick Reference

Ribbon

Click Output tab ➤ Publish panel ➤ Publish To Google Earth 🌐

Menu

File ➤ Publish To Google Earth

Command Line

PublishKML

PublishKMZ

Dialog Boxes

# Attaching Time Information to Model Data

The GETime command enables you to attach the timespan information to civil object entities. When you import civil model data that contains the timespan information to Google Earth, you can view the information sequentially or as an animation.

You can attach time-related information to each piece of the model data and then display these data based on start and end times (timespan). This command is useful when presenting project development over time.

In Google Earth, you can use the Time slider in Google Earth to control the display of the model data. You can, for example, redefine the time range or move the time range to a different point on a timeline. For more information on Google Earth timeline playback controls in the 3D Viewer, see "Viewing a Timeline" in the *Google Earth User Guide*.

Before you assign a timespan to a civil model, decide on the best way to associate time information with the various entities. For example, you can assign a timespan to each stage of a land development project from modeling a subdivision layout to grading work, or building a road. Alternatively, you can show the progress of each individual stage separately.

**To attach time information to an object**

**NOTE** If you plan to publish your model to Google Earth in the same session, ensure that Google Earth is running.

1   In the drawing, do one of the following:

■   Select an object. Click Surface tab ➤ General Tools panel ➤ Google Earth Timespan 🕒 .

■   At the command line, enter GETime.

**2** Select an object to assign timespan information and press Enter.

**3** In the Timespan for Google Earth dialog box (page 1873), specify the start and end dates by selecting them from the calendar. Click OK.

**4** Repeat steps 2 to 3 to assign timespan information to any other objects.

---

**TIP** Hover over an object to see a tooltip with timespan information attached to the model space entity.

---

**5** When finished, publish the model to Google Earth. See Publishing Civil Data to Google Earth (page 1669).

You can view the published model in the Google Earth 3D Viewer and adjust the necessary parameters using the timeline controls. See "Viewing a Timeline" in the *Google Earth User Guide*.

When you attach timespan information to Civil models, note the following:

■ If you do not attach the start-time information to an object entity, Google Earth starts the playback from the beginning of the timeline.

■ If you do not attach the end-time information to an object entity, Google Earth starts the playback from the start-time point and continues up to the end of the timeline.

■ An object entity without any timespan information attached is displayed in Google Earth constantly.

■ When publishing an object to Google Earth, you can not attach time information to this object and associate a render material with it at the same time.

### Quick Reference

Ribbon

Select an object. Surface tab ➤ General Tools panel ➤ Google Earth Timespan

Command Line

GETime

Dialog Box

Timespan for Google Earth (page 1873)

# Google Earth Import and Export Command Reference

You can use commands to quickly access Google Earth import and export functionality.

The following table lists the AutoCAD Civil commands related to import and export operations with Google Earth and briefly describes their functionality.

| Command | Description |
| --- | --- |
| GETime | Attaches the timespan information to civil object entities (page 1670) |
| ImportGEData | Imports Google Earth image and terrain data into a created TIN surface (page 1667) |
| ImportGEImage | Imports a Google Earth Image into a drawing (page 1665) |

| Command | Description |
| --- | --- |
| ImportGEMesh | Imports elevation data of the current Google Earth view into AutoCAD Civil as a mesh (page 1668) |
| ImportGESurface | Imports Google Earth terrain data into a created TIN surface (page 1666) |
| PublishKML | Publishes Civil design data to Google Earth in the KML (uncompressed) format (page 1669) |
| PublishKMZ | Publishes Civil design data to Google Earth in the KMZ (compressed) format (page 1669) |

# LandXML Import and Export

# 41

Use the AutoCAD Civil LandXML import and export commands to import and export drawing data in LandXML format.

See Understanding LandXML Import and Export (page 1673) for a list of objects that are supported in AutoCAD Civil 2010.

The LandXML import and export functionality is based on the LandXML schema.

AutoCAD Civil supports the following LandXML schema versions:

- LandXML-1.0

- LandXML-1.1

- LandXML-1.2

For more information about the LandXML schema, go to *www.landxml.org*.

## Understanding LandXML Import and Export

LandXML provides a non-proprietary data standard that is driven by an industry consortium of partners.

Use LandXML to transfer data to another drawing or to another application that supports imported XML. For example, you can use LandXML to transfer surfaces between drawings.

Many AutoCAD Civil objects are supported for LandXML import and export. See Supported LandXML Data for Drawing Import and Export (page 1674)for a list of supported objects.

In addition, Survey database data can be imported and exported. For more information, see Importing Survey XML Data.

By transforming AutoCAD Civil drawing data to LandXML, you can:

- **Exchange data**. Import LandXML data into other software applications. The data can then be modified and delivered to customers and agencies in the required formats.

- **Transfer or archive data**. Transfer data to another AutoCAD Civil drawing. Also, you can archive data in a non-proprietary format.

- **Convert units**. Export data using imperial measurements. Then, import it using metric to scale and convert values.

■ **Translate/rotate coordinates**. Globally adjust the elevations of data.

# Supported LandXML Data for Drawing Import and Export

LandXML supports AutoCAD Civil drawing data for import and export.

### General Data Handling

All coordinate locations are always treated as Northing, Easting, Elevation (or Y,X,Z).

All station values are treated as the actual measured distance along the alignment or geometry. Station equations are always imported or exported (if they are defined) with alignment data. However, they are not applied to any geometric station locations in the data; they are used only for display and reporting purposes.

Point references to pointType derived locations <CgPoint pntRef="100"/> are supported. The supported references include: CgPoint, Start, Center, End, Monument, and CrossSectPnt elements. For more information, see .

### LandXML Elements Supported for Drawing Import

| LandXML Element Supported | Notes |
| --- | --- |
| Units | |
| Application | |
| Author | |
| CoordinateSystem | Uses Map Zone name / European Petroleum Survey Group (EPSG) name |
| **CgPoints** | Supports multiple elements. If <CgPoints> is named, a corresponding point group is created and COGO points are added. |
| CgPoint | Imports a COGO point and uses name and desc or code attributes. Number is auto-assigned if name is alpha-numeric, but name is maintained. |
| **Alignments** | |
| CoordGeom | Lines, curves, spirals, and irregular lines. |
| AlignPI | |
| StaEq | |
| Superelevation | |
| DesignSpeeds | |

| LandXML Element Supported | Notes |
|---|---|
| Profiles | Finished ground, design profiles created using imported lines (PVIs), circular vertical curves, parabolic, symmetric, and asymmetric vertical curves. |
| ProfAlign | All elements supported |
| ProfSurf | PVI, pointlist data for a sampled ground surface |
| CrossSects | |
| CrossSectSurf | As sample lines in sample line group associated with alignment |
| DesignCrossSectSurf | As subassembly |
| **Parcels** | Creates parcels from data |
| CoordGeom | All elements supported |
| **Monuments** | Creates COGO points in a monuments points group |
| **Surfaces** | |
| SourceData | |
| Breaklines | Imported as 3D polylines on layer "<surface-name>_Breaklines" |
| Contours | Imported as 3D polylines on layer "<surface-name>_Contours" |
| DataPnts | Imported as COGO point group named "<surfa-cename>_DataPoints" on layer "<surface-name>_DataPoints" |
| Definition | |
| TIN | Option to import faces, then either update trian-gulation or maintain the triangulation in the file. Supports face edge visibility and face neighbor optimization attributes. |
| GRID | |
| **Survey** | |
| ReducedObservations | Creates COGO points in an observations point group. |

| LandXML Element Supported | Notes |
| --- | --- |
| | **IMPORTANT** Using the Import LandXML command from the File menu does not import survey data into the survey database. For more information, see Importing Survey XML Data. |
| SurveyMonuments | Creates COGO points in a monuments point group |
| **PlanFeatures** | |
| CoordGeom | Imported as 3D polylines to current layer |
| **PipeNetworks** | Each <Pipe Network> is imported as a AutoCAD Civil pipe network |
| Pipe | Supports curved pipe |
| Units.diameterUnit | |
| Structs | |
| Units.diameterUnit | |
| PipeFlow | |
| StructFlow | |

**Supported LandXML Elements for Drawing Export**

| LandXML Element Supported | Notes |
| --- | --- |
| Units | Always exported |
| Application | Always exported |
| Author | |
| CoordinateSystem | Uses Map Zone name / European Petroleum Survey Group (EPSG) name |
| **CgPoints** | Point group exports as <CgPoints> with matching name and all points export. If an AutoCAD Civil point has a point name, the name is exported to the LandXML point name attribute. Otherwise, the point number is exported to the LandXML point name attribute. |
| **Alignments** | |
| CoordGeom | As lines, curves, and spirals. |
| AlignPI | |

| LandXML Element Supported | Notes |
| --- | --- |
| StaEq | |
| Superelevation | |
| DesignSpeeds | |
| Profiles | |
| ProfAlign | Design profiles with lines, circular vertical curves, parabolic, symmetric, and asymmetric vertical curves. |
| ProfSurf | |
| CrossSects | |
| CrossSectSurf | Sample lines exported as CrossSectSurfs |
| DesignCrossSectSurf | Assemblies exported as DesignCrossSectSurfs |
| **Parcels** | |
| CoordGeom | As lines, curves, and spirals. |
| **Surfaces** | TIN, GRID, and volume surfaces exported. |
| SourceData | |
| Breaklines | |
| Contours | |
| DataPnts | |
| Definition | |
| TIN | |
| GRID | |
| **Roadways** | Corridors exported as <Roadways> with references to <Alignments> with Profiles and Cross Sections (both sampled ground and design sections) and to one or more reference surfaces. |
| **PipeNetworks** | |
| Pipes | |
| Units.diameterUnit | |
| Structs | |

| LandXML Element Supported | Notes |
|---|---|
| Units.diameterUnit | |
| PipeFlow | |
| StructFlow | |

## LandXML and Sites

In an AutoCAD Civil drawing, a site is used to collect or group sub-collections, such as parcels, alignments, and gradings, by a topology that they share.

When objects that are grouped under the Sites collection are exported to a LandXML file, the site name is exported with them.

When objects are imported from a LandXML file into the drawing, two site pickers are displayed:

■ Alignments Site allows you to specify the site on which the alignments will reside. The default selection is <none>, which places the alignments in the top-level Alignments collection and prevents them from interacting with parcels.

■ Parcels Site allows you to specify the site on which the parcels will reside. If no sites exist in the drawing, the parcels are placed on the default site (Site 1). If the LandXML file has a site name for its features, then that site name is created in the drawing and the parcels are assigned to it.

For more information about sites, see Understanding Sites (page 719).

## Handling Duplicate LandXML Data on Import

When objects in the drawing have identical names to objects in a LandXML file, the imported data can be skipped, renamed upon import, or it can overwrite the existing data.

You can specify your preference for handling duplicate data under Conflict Resolution Settings on the Import tab of the LandXML Settings dialog box (page 1978).

# Viewing and Editing LandXML Drawing Settings

You can view and edit LandXML settings for the entire drawing.

These settings control how data is converted between AutoCAD Civil and a LandXML file, including coordinate translation and rotation, and description information.

In the Toolspace, on the Settings tab, right-click the drawing collection, then, click Edit LandXML Settings. Use the LandXML Settings (page 1978) dialog box, to view and edit all of the LandXML settings for the drawing.

For more information about using the Toolspace and the Settings tab, see The Toolspace Window (page 80).

## Viewing and Editing the LandXML Import Settings

Control how LandXML data is inserted into the drawing when the file is imported.

**To view and edit LandXML import settings**

1 In Toolspace, on the Settings tab, right-click the <drawing name> collection ➤ Edit LandXML Settings.

2 In the LandXML Settings dialog box, click Import tab (page 1978) and modify the settings.

**3**  Click OK.

The settings are saved as the default settings for the drawing.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <drawing name> ➤ Edit LandXML Settings

Dialog Box

LandXML Settings (page 1978)

## Unit Conversion for LandXML Data

AutoCAD Civil uses conversion factors if the LandXML file you are importing has different units from the current drawing.

You can import LandXML files that use feet or meters.

When imported, the LandXML files are converted to the units of the current drawing. In AutoCAD Civil, the default drawing units can be meters, International Feet, or U.S. Survey Feet. For more information, see Specifying Units and Zone Settings (page 63).

**NOTE** Previously, it was required that you specify International or US Survey Foot in the LandXML settings. This specification is now done in the drawing settings.

**NOTE** Unit conversion is not applied to descriptive LandXML attributes that may contain numeric values. For example, if you are importing a point with the description "12 oak" into a metric project, it will not be changed to "305mm oak."

## Translating and Rotating LandXML Data

Translate imported coordinate data by specifying translation and rotation settings. Use the Import tab of the LandXML Settings dialog box to set the values.

■ **Base point values**. The base point coordinates in the drawing are used to translate and rotate all imported values. These coordinates are in the drawing units.

■ **Translate coordinate values**. The coordinate values are used to translate data from the base point. These coordinates are in the drawing units.

**NOTE** In certain circumstances, you can assign the same values to both the translated coordinates and the base point coordinates. For example, the values would be the same if you want to rotate the coordinates without translating them.

For example, the following formula is used to translate the imported elevation values:

(Translation Coordinate Elevation - Base Point Elevation) + LandXML Elevation = Imported elevation value

Therefore, if the LandXML file elevation is 90 feet, the base point elevation is 0.000 feet, and the translate coordinate elevation is 10.000 feet, the formula is:

(10.000 - 0.000) + 90 = 100

■ **Rotation angle and direction**. The angle and direction, measured about the base point, about which the data is rotated.

## Viewing and Editing the LandXML Export Settings

Control how data in AutoCAD Civil is exported to a file in LandXML format.

**To view or edit LandXML export settings**

1   In Toolspace, on the Settings tab, right-click <drawing name> ➤ Edit LandXML Settings.

2   In the LandXML Settings dialog box, click the Export tab (page 1981) and modify the settings.

For more information, see Point Reference Export Options (page 1680) and Point Description and Code Matching (page 1680).

3   Click OK.

The settings are saved as the default settings for the drawing.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click <drawing name> ➤ Edit LandXML Settings

Dialog Box

LandXML Settings (page 1981)

## Point Reference Export Options

When selecting the data to export, you can choose to export point references. Set the Export Point References property on the Export tab (page 1981) of the LandXML Settings dialog box to On.

When you turn this option on, AutoCAD Civil creates COGO point references, where possible, for the parcel, alignment, and surface geometry. This means that, if the geometry of parcels, alignments, and surfaces in the drawing match COGO point coordinates within a specified tolerance, the object coordinates are exported to the LandXML file as references to those COGO points.

Point references substitute a known point name instead of using northing and easting coordinates. For example, without a point reference, the start point of a line element is written as follows:

```
<Start>5447.73530 4525.60643</Start>
```

With a point reference, it is written as follows:

```
<Start pntRef="371"></Start>
```

It is also valid for a coordinate geometry element to have a mix of pntRef and coordinate values as shown in the following example:

```
<Line> <Start pntRef="250" /> <End>5632.87775298 3944.16966215</End> </Line>
```

This situation could occur for the endpoint if a COGO point does not exist within the specified tolerance.

## Point Description and Code Matching

Match the AutoCAD Civil description information with LandXML attributes. Use the Code Attribute and Desc Attribute property groups on the Export tab of the LandXML Settings dialog box.

You can either disable or match the 'code' attribute in LandXML to a point's description:

■   **Disabled**: Does not export a point description to the 'code' attribute.

- **Raw Description**: Exports a point raw description to the 'code' attribute.

- **Full Description**: Exports a point full description to the 'code' attribute.

---

**NOTE** Typically, you would export the raw description to the 'code' attribute.

---

You can either disable or match the 'desc' attribute in the LandXML to a point's description:

- **Disabled**: Does not exports a point description to the 'desc' attribute.

- **Raw Description**: Exports a point raw description to the 'desc' attribute.

- **Full Description**: Exports a point full description to the 'desc' attribute.

---

**NOTE** Typically, you would export the full description to the 'desc' attribute.

---

You can also specify whether to export the full description if it is the same as the raw description. For example, if the raw and full descriptions are the same, then only one value is exported. This would occur when points do not have defined description keys.

# Importing LandXML Drawing Data

Import LandXML data into the current AutoCAD Civil drawing.

Importing LandXML data into AutoCAD Civil is a two-step process:

1. Specify the import settings that prepare the data for import. For more information, see Viewing and Editing LandXML Drawing Settings (page 1678).

2. Select the files and the specific data in the file to import.

The LandXML import functionality automatically handles the conversion between the units specified in the LandXML file and the current drawing units.

---

**NOTE** LandXML does not transform coordinate systems automatically. No specific coordinate system transformation is applied other than what is specified by the translation and rotation settings (page 1679).

---

**To import LandXML data**

1. Click Insert tab ➤ Import panel ➤ LandXML 🗃 .

2. In the Import LandXML dialog box, select or browse to the LandXML (*.xml*) file you want to import. Click Open.

3. In the Import LandXML dialog box (page 1977), select sites for the alignments and parcels. See LandXML and Sites (page 1678) for more information.

4. The data tree displays each of the major data collections in the LandXML file. Use the data tree to:
   - Navigate a data collection and view its subcomponents.

   - Expand the collections. Select or clear the check boxes to filter the data types that you want to import into the drawing. By default, all data components are selected for import.

---

**NOTE** The check boxes have a tri-state display. If only some objects are selected under a collection, the check box is shaded ☑ . If all the items are selected, the check box is selected ☑ . If all the items below the collection are cleared, the check box for the collection is cleared ☐ .

---

**5** Click OK to import the LandXML file or click Browse to select another LandXML file.

After you click OK, the data is imported into the drawing. The data components are added to the drawing's applicable data collections in the Prospector tree.

**NOTE** As each object is imported, the Event Viewer Vista (page 2455) dialog box can display the status of each operation. For more information, see The Event Viewer Vista (page 1626).

### Quick Reference

Ribbon

Insert tab ➤ Import panel ➤ LandXML

Menu

File menu ➤ Import ➤ Import LandXML

Command Line

LandXMLIn

Dialog Box

Import LandXML (page 1977)

# Exporting LandXML Drawing Data

Export LandXML drawing data from AutoCAD Civil.

You can select the Export LandXML command from the File menu, or you can right-click an object collection in the Prospector tree and select the command.

When you export from the Prospector tree, the Export To LandXML dialog box contains a pre-defined selection set of that collection's data. When you export from the File menu, all drawing data is selected by default, and you can adjust the selection by clearing check boxes. You can alternately specify the objects to export by selecting them from the drawing.

**To export LandXML data**

**1** Click Output tab ➤ Export panel ➤ Export To LandXML .

The Export To LandXML (page 1983) dialog box, which contains a data tree, is displayed. In the tree, each of the major data collections is available for export to the LandXML file.

**2** Do one of the following:

- Select or clear the check boxes to filter the data types that you want to export to the LandXML file. Then click OK.

  **NOTE** The check boxes have a tri-state display. If only some objects are selected under a collection, the check box is shaded . If all the items are selected, the check box is selected . If all the items below the collection are cleared, the check box for the collection is cleared .

- Select data from the drawing by clicking and selecting the objects in the drawing. Press *Enter* after you finished selecting the objects.

**NOTE** When you select one point in a point group, the entire point group is selected. If the point belongs to more than one point group, the point group with the highest display order is selected. For more information, see Changing the Point Group Display Order (page 526).

**3** Specify the LandXML schema version.

**4** In the Export To LandXML dialog box, enter the name of the LandXML (*.xml*) file you want to export and specify the location. Click Save.

### Quick Reference

Ribbon

Output tab ➤ Export panel ➤ Export To LandXML

Menu

File menu ➤ Export ➤ Export To LandXML

Toolspace Shortcut Menu

Prospector tab: Right-click <collection name> ➤ Export LandXML

Command Line

LandXMLOut

Dialog Box

Export To LandXML (page 1983)

## LandXML Command Reference

The AutoCAD Civil commands for LandXML and a brief description of their functionality. For more information, follow the links.

| Command | Description |
| --- | --- |
| LandXMLIn | Imports LandXML data into AutoCAD Civil (page 1681) |
| LandXMLOut | Exports AutoCAD Civil data to LandXML (page 1682) |

# Importing Architectural Data

# 42

Use the Import Building Site command to import architectural model data from Autodesk Revit Architecture to AutoCAD Civil in a collaborative design project.

By implementing the seamless data exchange between Autodesk Revit Architecture and AutoCAD Civil, a site design team can

- Finalize the site plans before the architectural design has been completed.

- Obtain enough information to start preparing the site for construction.

- Reduce the communication cycle between architectural and civil engineering teams.

🔊 Importing Architectural Data Workflow

## Publishing a Design Package File

Use Autodesk Revit Architecture to prepare and publish an architectural model package file.

An architect prepares the architectural model in Autodesk Revit Architecture, applies the corresponding templates, visibility settings and filters, and then publishes the model to the design package file (.adsk). The imported model represents a gross area footprint of the building, with property and utility connection points.

The design package file serves as a medium for sharing data between Autodesk Revit Architecture and AutoCAD Civil and contains the following elements:

- An image of a building footprint

- Data for the building footprint (for example, area and perimeter)

- Images of building model categories (doors, floors, roofs) visible in specific views

- Data for the building model categories (for example, slope and area for the Roof model category)

- An image of a site model

- Base point coordinates data (Easting, Northing, and elevation)

- Property line graphics

- Project information data (for example, building type, occupancy, and total gross building area)
- Package file metadata for workflow type, OPC, and utilities

The following high-level procedure is performed by an architect in Autodesk Revit Architecture to prepare the architectural model for sharing with AutoCAD Civil.

**To publish a design package file in Autodesk Revit Architecture**

- Prepare and simplify the file. The user applies a Civil Engineering View template and related visibility settings and filters.
- Launch the Export Module and specify export settings. The user specifies a footprint level, location, base point, and the number of building model elements to export.
- Save the file. The user specifies the file location and file name.
- View the Export Building Site report that lists a summary of the output contents.

For details, see Exporting Building Sites in the *Autodesk Revit Architecture 2010 User's Guide*.

# Preparing for Importing a Design Package File

Close interaction of architects and civil engineers is important when sharing data between Autodesk Revit Architecture and AutoCAD Civil.

The following is a summary of issues that you should be aware of and steps that you can perform to facilitate the collaboration effort:

- When preparing a design package file for export, the architect can create site utilities with host connectors that are functional in AutoCAD Civil. You should make the architect aware of the utilities that are relevant to the civil project beforehand, so they are exported with the design package file. If the required utilities are not supported in Revit Architecture, the architect would be able to notify you. If you prefer to set up the utility connections yourself, ask the architect to omit the Utilities parameter in the export settings.
- Before finalizing the building site model export, the architect may contact you about the site survey point coordinates, so they can be exported with the model file.
- The architect should apply as much model simplification settings as possible. You should communicate to the architect your minimum requirements regarding the design package file content.
- If the import operation is slow, you can request the architect to reexport the model with modified settings to reduce file size.
- The Export Building Site report is saved with the ADSK building site model file. You can use the report to compare the content of the original file with the imported file.
- Establish a procedure when every time the source design package file is modified, the architect notifies you immediately, so you can update the building site object in your drawing.

# Using the Wizard to Import a Building Site Object

Use the Import Building Site wizard to bring the architectural model, created in Autodesk Revit Architecture, into AutoCAD Civil, confirm and compare the building site units the Civil drawing units, and specify the object insertion point.

**To import a building site object**

1   Click Insert tab ➤ Import panel ➤  ➤ Import Building Site  .

2   On the Select File page of the Import Building Site wizard, click  and select the published ADSK file to import.

   The model image appears in the Preview window. The  marker shows the base point specified in the building site object.

3   Click  to select or modify a template for the building site object name.

4   Optionally, use the ViewCube to manipulate the object in the Preview window. See Overview of ViewCube in the *AutoCAD User's Guide*.

5   Click Next.

   **NOTE** You can click Finish at any time to close the wizard and finish the import.

6   On the Confirm Units page, compare the units used in the building site object, as defined in the ADSK file, with the units used in the current AutoCAD Civil session. All the controls on this page are read-only. Use them for reference information.

7   Click Next.

8   On the Display Properties page, click  to specify the object layer (page 1985).

9   Optionally, click  to edit the default building site style. See Editing a Building Site Style (page 1688).

10   Click Next.

11   On the Insertion Point page, confirm and modify the coordinates and rotation angle of the building site object base point or click Specify In Drawing to orient the model in the drawing manually.

12   Optionally, click Preview to insert the building site object into the drawing as a test.

13   Do one of the following:
   ■   If you are in preview mode, press *Esc* to return to the Import Building Site wizard and continue fine-tuning the base point location.

   ■   Press *Enter* to close the Import Building Site wizard and leave the building site object in the drawing.

14   Click Finish.


## Quick Reference

Ribbon

Click Insert tab ➤ Import panel ➤  ➤ Import Building Site  .

Command Line

ImportBuildingSite

Dialog Box

Import Building Site wizard (page 1794)

# Creating a Building Site Style

Use the building site style to determine the display properties of the building site object components, such as building footprint, property lines, utilities, site model, and building model.

Use the Toolspace Settings tree to create a building site style.

**To create a building site style**

1   In Toolspace, on the Settings tab, right-click the Building Site Styles collection and click New.

2   On the tabs of the Building Site Style (page 1796) dialog box, enter the required data or specify the settings.

3   Click OK.

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: Building Site ➤ right-click Building Site Style ➤ Edit

Dialog Box

  Building Site Style (page 1796)

# Editing a Building Site Style

Edit the building site style that determines the display properties of the building site object components, such as building footprint, property lines, utilities, site model, and building model.

Use the Toolspace Settings tree to edit a building site style.

**To edit a building site style**

1   In Toolspace, on the Settings tab, right-click the name of the building site style that you want to edit and click Edit.

2   In the Building Site Style (page 1796) dialog box, change the properties of the building site style.

3   Click Apply and then OK.

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: Building Site ➤ Building Site Styles ➤ right-click <style name> ➤ Edit

Command Line

  EditBuildingSiteStyle

Dialog Box

  Building Site Style (page 1796)

# Modifying a Building Site Object in the Drawing

Use the EditBuildingSite command to edit the properties of the building site object in the drawing.

**To edit the building site object properties**

1   In Toolspace, on the Prospector tab, right-click a Building Site item and select Building Site Properties.

2   In the Building Site Properties (page 1798) dialog box, modify the name of the building site object, enter or edit the description, and modify the style.

3   Click Apply and than OK.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Building Site ➤ right-click <building site name> ➤ Building Site Properties

Command Line

EditBuildingSiteStyle

Dialog Box

Building Site Properties (page 1798)

# Updating a Building Site Object Definition

Use the UpdateBuildingSite command to update the building site object that was imported into the drawing.

You use this command if the source architectural model has changed and a new ADSK package file was

posted in the shared location. When changes are made to the package file, the out-of-date icon   displays next to a building site item in the Prospector tree. You can update the building site object references without reimporting the object into the drawing.

**To update a building site definition**

1   In Toolspace, on the Prospector tab, right-click a Building Site item and select Update Building Site Definition.

2   In the Select Autodesk Exchange File dialog box, select the updated ADSK file.

3   Click Open.

The definition of the imported building site object is updated and all its references are redefined with new data.

## Quick Reference

Toolspace Shortcut Menu

Prospector tab: Building Site ➤ right-click <building site name> ➤ Update Building Site Definition

Command Line

UpdateBuildingSite

# Command Reference

You can use commands to quickly access the functionality of the Building Site object.

The following table lists the AutoCAD Civil commands that relate to editing, importing, and updating the Building Site object, and briefly describes their functionality:

| Command | Description |
| --- | --- |
| EditBuildingSite | Opens the Building Site Properties dialog box, where you can modify the properties of the building site reference object (page 1688) |
| EditBuildingSiteStyle | Opens the Building Site Style dialog box where, you can edit the building site style (page 1688) |
| ImportBuildingSite | Imports a building site object into a Civil drawing (page 1686) |
| UpdateBuildingSite | Updates a building site object definition when the original building site object data were changed (page 1689) |

# Plan Production Tools

# 43

Use AutoCAD Civil plan production tools to quickly create construction documents from drawings.

## Understanding Plan Production Tools

AutoCAD Civil plan production tools automate the process of creating construction documents from your designs.

Use the plan production tools to quickly create sheets that automatically display sections of alignments and profiles in your plans.

Instead of having to manually create viewports along an alignment, and manually recreate sheets each time your data changes, you can create view frame groups that automatically capture predefined areas along an alignment. Select an alignment in your drawing and in just a few minutes you can create sheets that automatically display the desired data.

The plan production tools include the following components:

■ **Plan Production commands**. You can access the plan production commands from the Output tab and Plan Production contextual tab on the ribbon. The Plan Production contextual tab is displayed when you select a view frame or a match line in the drawing.

■ ⬚ **Create View Frames Wizard**. Start the process of using the plan production tools by using this wizard to define a group of view frames along an alignment. For information, see Creating View Frames (page 1703).

■ ⬚ **View Frame Group**. The view frame group object helps you manage a single group of view frames that are displaying consecutive station ranges along the same alignment. You can set many options at the view frame group level, such as styles and labeling. View frame group objects are displayed in the Prospector tree, and you can control their default command settings in the Settings tree. For information, see The View Frame Group Object (page 1694).

■ ⬚ **View Frames**. View frames are rectangular-shaped regions along an alignment that define an area that will be displayed in a sheet. The view frame size, shape, and scale comes from a designated viewport that exists on the layout tab of a specified template. After view frames are created, the properties of the view frame objects are saved in the currently open drawing. The view frame objects are displayed in the drawing and in the Prospector tree, and you can control their default style and labeling in the Settings tree. For information, see The View Frame Object (page 1695).

- ▣ **Match Lines**. In the AutoCAD Civil plan production features, a match line is a straight line that indicate locations in a view frame group where one view frame intersects or matches up with another view frame. Match lines are only displayed in paper space and only in plan view. They are designed to visually indicate the locations (start and end stations) along an alignment where each view frame begins and ends. Match lines have their own object style and they typically include labels that can identify both the previous and next sheet (view frame) along an alignment. You have the option to include a left side match line label, a right side match line label, both, or none, and you can choose where along the match line you want the label to be displayed (top, middle, end of match line). Like view frame objects, match line objects are also displayed in the Prospector tree, and you can control their default style and labeling in the Settings tree. For information, see The Match Line Object (page 1696).

- ▣ **Create Sheets Wizard**. After you have used the Create View Frames wizard, the next step is to use the Create Sheets wizard to quickly create your sheets. For information, see Creating Sheets (page 1719).

After you have created view frames and sheets, you can then use the Sheet Set Manager to organize, publish, share, and manage sheet sets. For information, see "Work with Sheets in a Sheet Set" in the AutoCAD Help.

# Prerequisites for Using Plan Production Tools

A few prerequisites must be in place before you can use the plan production tools.

You can use the plan production tools if you have a drawing that contains an alignment, and if you have access to a template that contains certain types of preconfigured viewports.

Because there are three different categories or types of sheets that you may want to create (plan and profile, plan only, or profile only), the prerequisites are slightly different depending on the type of sheets you want to create.

The simplest way to get started using the plan production tools is to create "plan only" sheets. As long as the current drawing contains an alignment, and you can access the Plan Only template supplied with AutoCAD Civil (which is named *Civil 3D (Imperial) Plan only.dwt*), you can successfully use both wizards. If the current drawing does not contain an alignment, or if you do not have access to the correct type of template, you will not be able use the Create View Frames wizard.

The following table summarizes the prerequisites for using plan production tools:

| Sheet type | Prerequisites for using plan production tools |
|---|---|
| Plan Only | ■ The current drawing must contain an alignment.<br><br>■ You must be able to access a template that contains a viewport with a Viewport Type defined as Plan, such as the *Civil 3D (Imperial) Plan Only.dwt* template located in the Template\Plan Production folder. |
| Profile Only | ■ The current drawing must contain an alignment and a profile.<br><br>■ You must be able to access a template that contains a viewport with a Viewport Type defined as Profile, such as the *Civil 3D (Imperial) Profile Only.dwt* template located in the Template\Plan Production folder. |
| Plan and Profile | ■ The current drawing must contain an alignment and a profile. |

| Sheet type | Prerequisites for using plan production tools |
|---|---|
| | ■ You must be able to access a template that contains a viewport with a Viewport Type defined as Plan, and a viewport with a Viewport Type defined as Profile, such as the *Civil 3D (Imperial) Plan and Profile.dwt* template located in the Template\Plan Production folder. |

**NOTE** When you install AutoCAD Civil, the sample plan production templates are installed to the following location by default: *C:\Documents and Settings\<user name>\Local Settings\Application Data\Autodesk\Civil2010\enu\Template\Plan Production*.

For more information about setting the Viewport Type in a template, see Configuring Viewports for Plan Production (page 1700).

For step-by-step information about how to use the wizards, see Creating View Frames (page 1703) and Creating Sheets (page 1719).

# Plan Production Tools Object Styles and Display

Object styles control how plan production tool objects (view frames and match lines) are displayed.

The following illustration shows the view frame and match lines collections in the Settings tree:



The following table describes the display styles available for the plan production tools components:

| Plan Production Tool Object | Display and labeling styles available |
|---|---|
| View Frames | View frame style and view frame label style |
| Match Lines | Match line style and match line label style |
| View Frame Group | View frame groups are non-graphical objects that manage a collection of view frames and match lines. At the view frame group level, you can set the display and label style defaults for view frames and match lines. |

For more information about styles, see Object Styles (page 53), View Frame Style dialog box (page 2098) and Match Line Style dialog box (page 2099).

For more information about labeling for view frames and match lines, see Labeling View Frames (page 1713), and Labeling Match Lines (page 1718).

Before creating view frames and view frame groups, you should be familiar with creating and managing styles (page 53).

The top-level plan production tool object, the view frame group object, has command-level settings that can be configured from the Settings tree. For general information about command settings, see command settings (page 97).

**See also:**

■  Edit Feature Settings - View Frame Group Dialog Box (page 2081)

■  Edit Feature Settings - View Frame Dialog Box (page 2084)

■  Edit Feature Settings - Match Line Dialog Box (page 2084)

# The View Frame Group Object

View frame groups persist in an AutoCAD drawing as objects with the name AeccViewFrameGroup.

A view frame group object manages a group of view frames that are created while using the Create View Frames wizard. When you use the Create View Frames wizard, you are prompted to enter a name and other object creation criteria for a view frame group. Each view frame group manages the view frames and match lines for a single alignment.

**See also:**

■  Understanding Objects and Styles (page 51)

# View Frame Groups Collection (Prospector Tab)

Use the View Frame Groups collection in the Prospector tree to access the view frame groups in a drawing or a project.

After View frame group objects are created, they are displayed in the View Frame Groups collection in the Prospector tree.

When you expand a view frame group in Prospector, you can see the view frames and match lines associated with it. Expand the View Frames or Match Lines collections to view the names of the view frames or match lines in that group. Selecting an individual view frame or match line in Prospector displays a tabular list of the view frame and match line data in the Prospector list view. For more information, see The Toolspace Item View (page 82).

# View Frame Group Collection (Settings Tab)

Use the View Frame Group collection in the Settings tree to manage view frame group settings and command settings.

Right-click the View Frame Group collection to do the following:

■  Edit the view frame group feature settings.

■  Edit the view frame group label style default settings.

■ Refresh the display of the settings tree.

For more information, see "The Object Collection (Settings Tree)".

Expand the View Frame Group collection to display and edit the default styles and command settings that are available for view frame groups.

| For information about... | See... |
|---|---|
| View frame group settings | Edit Feature Settings - View Frame Group Dialog Box (page 2081) |
| View frame group styles | Plan Production Tools Object Styles and Display (page 1693) |

# The View Frame Object

View frames persist in an AutoCAD drawing as objects with the name AeccViewFrame.

View frame objects are created automatically when you use the Create View Frames wizard. View frames are always associated with a parent view frame group object.

View frames are created based on an alignment in the drawing, and on a designated plan view or profile view viewport in a template. When you choose to create sheets that will contain profile view data in addition to plan view data, you must also have a profile in the drawing. The properties of the view frame are saved in the drawing in which you create the view frame.

Most of the view frame object information is derived from the viewport information specified in the template. For example, the view frames you create get their size and scale from the viewport in the template. Similarly, the sheet layout is also derived from the referenced template.

An important concept to understand is that the viewports in the referenced template must have a property called Viewport Type set to either plan or profile, depending on the type of data you want that viewport to display. For more information, see Configuring Viewports for Plan Production (page 1700).

**See also:**

■ Understanding Objects and Styles (page 51)

# View Frame Styles and Display

How view frames are displayed is controlled by a view frame style, a view frame label style, and choices made on the Create View Frames wizard.

**View Frame Style**

The view frame style controls the appearance of the view frame. You can set typical style characteristics, such as visibility, layer, color, linetype, linetype scale, and lineweight.

View frames have only one display component, the border, which is visible only in plan view.

For more information, see View Frame Style Dialog Box (page 2098) or Object Styles (page 53)

**View Frame Label Style and Location**

You can specify a label style and a location for the label that will be displayed on each view frame. Set the defaults for this view frame label behavior using the View Frame Group Settings dialog box.

You can choose from a variety of locations around the view frame to place the view frame label. For example, you can choose to place view frame labels at the top left, top center, middle right, and so on. You make this choice in the Create View Frames wizard. You can change this location later, after it has been created by right-clicking the label and selecting Edit Label. For more information, see Labeling View Frames (page 1713).

## View Frames Collection (Prospector Tab)

Use the View Frame Groups collection in the Prospector tree to access the view frames in a drawing or a project. As view frame objects are created, they are displayed in the View Frame Groups collection.

Expand the View Frame Groups collection, and then expand a view frame group object beneath it to view the names of the view frames associated with a view frame group. To display a tabular list of the view frames in the Prospector list view, select a view frame in the Prospector tree. For more information, see The Toolspace Item View (page 82).

## View Frame Collection (Settings Tab)

Use the View Frame collection in the Settings tree to manage view frame feature settings, styles, and command settings.

Right-click the View Frame collection to do the following:

■ Edit the view frame feature settings.

■ Edit the view frame label style default settings.

■ Refresh the display of the settings tree.

For more information about the contents of this shortcut menu, see "The Object Collection (Settings Tree)".

Expand the View Frame collection to display and edit the styles that are available for view frames.

| For information about... | See... |
|---|---|
| View frame settings | Edit Feature Settings - View Frame Dialog Box (page 2084) |
| View frame styles | Plan Production Tools Object Styles and Display (page 1693) |

## The Match Line Object

Match lines persist in an AutoCAD drawing as objects with the name AeccMatchLine.

If you selected the Insert Match Lines check box on the Match Lines page of the Create View Frames wizard, a match line is created at each view frame intersection. The first and last view frames in a group have only one match line. For plan only sheets, you have the option to not insert match lines on the view frames. To do this, simply remove the check mark from this check box on the Create View Frames wizard.

NOTE Match lines will always be inserted in profile views. Therefore, when creating plan and profile or profile only sheets, match lines will always be inserted, and the Insert Match Lines check box is selected but unavailable (you cannot remove the check mark). If you are creating plan only sheets, you can remove the check mark from the box and match lines will not be inserted in the resulting plan views.

Properties of match lines, such as where they are positioned on the alignment, a match line style, and a match line label style, can be specified on the Match Lines page of the Create View Frames wizard. You can set defaults for their styles using the Match Line settings available on the Settings tab in Toolspace.

Match lines have a match line mask area on the outside of each match line. This lets you mask data outside of the match lines. You can set hatch pattern options for this area using the Match Line Style dialog box.

Match lines and the match line mask area are displayed only in plan views, not in profile views. The match line mask area exists only in paper space.

**See also:**

- Editing the Match Line Mask Area (page 1715)
- Understanding Objects and Styles (page 51)

## Match Lines Styles and Display

How match lines are displayed is controlled by a match line style, a match line label style, and choices made on the Match Lines page of the Create View Frames wizard.

**Match Line Style**

Use the match line style to control the appearance of the match line. You can set typical style characteristics, such as visibility, layer, color, linetype, linetype scale, and lineweight. For more information, see Match Line Style Dialog Box (page 2099) or Object Styles (page 53).

**Match Line Mask Area**

Use the match line mask area to hide or mask the area that exists between the outside of a match line and the inside border or boundary of the view frame. This component exists only in plan views, not profile views, and only in paper space. The following illustration shows an example of the two match line mask areas in gray, on the left and right sides of a plan view viewport:



The Display tab of the Match Line Style dialog box includes an option to specify a hatch pattern for this match line mask area. Use this option to help you make your construction documents look better by masking the drawing data that exists outside of the match lines. See Editing the Match Line Mask Area (page 1715) for more information.

Match line labels are displayed on top of the match line mask area, and are therefore visible over it.

**Match Line Label Style and Location**

You can specify a label style for the label that will be displayed on the left side of a match line or on the right side of a match line. Since each individual match line is displayed in two sheets, it's convenient to have the ability to set label styles differently for the label that will be displayed on the left side of the match line, and another label style for the label that will be displayed on the right side of the match line.

Also, you can display only one match line label (on the left or on the right side of the match line) by simply turning off the visibility of one match line label.

You can choose to place the left and right side match line labels at the start, end, or middle of the match line, or where the alignment intersects the match line. For more information, see Labeling Match Lines (page 1718).

Set the defaults for this match line label behavior using the View Frame Group Settings dialog box.

# Match Lines Collection (Prospector Tab)

Use the View Frame Groups collection in the Prospector tree to access the match lines in a drawing or a project.

As match lines objects are created, they are displayed in the parent View Frame Groups collection they are associated with. Expand the View Frame Group collection, and then expand a view frame group object beneath it to view the names of the match lines associated with a view frame group. To display a tabular list of the match lines in the Prospector list view, select a match line in the Prospector tree. For more information, see The Toolspace Item View (page 82).

# Match Line Collection (Settings Tab)

Use the Match Line collection in the Settings tree to manage match line settings, styles, and command settings.

Expand the View Frame collection to display and edit the styles that are available for match lines.

Right-click the Match Line collection to do the following:

■ Edit the match line feature settings.

■ Edit the match line label style default settings.

■ Refresh the display of the settings tree.

For more information about the contents of this shortcut menu, see "The Object Collection (Settings Tree)".

| For information about... | See... |
| --- | --- |
| Match line settings | Edit Feature Settings - Match Line Dialog Box (page 2084) |
| Match line styles | Plan Production Tools Object Styles and Display (page 1693) |

# Labeling Plan Production Tool Components

By default, view frames and match lines contain standard object labels.

If you want view frames and match lines to be displayed without labels, you can delete the labels or turn off the visibility of the label.

The View Frame Group page of the Create View Frames wizard contains the option for setting the view frame labels, and the Match Lines page contains the option for setting match line labels.

In the drawing, there are no labels for view frame group objects. However, since the view frame group object is the parent object for view frames, you can set label defaults for view frames using by View Frame Group collection in the Toolspace Settings tree. Right-click View Frame Group in the Settings tree, and click Edit Label Style Defaults on the shortcut menu. The Edit Label Style Defaults - View Frame dialog box is displayed. For more information, see Edit Label Style Defaults Dialog Box (Feature Level) (page 1941).

On the Toolspace Settings tab, in the View Frame ➤ Label Style collection, set up label styles to use for view frames. Do the same for match lines using the Match Line ➤ Label Style collection.

**See also:**

■ Labeling View Frames (page 1713)

■ Labeling Match Lines (page 1718)

# Preparing for Plan Production

Set up your drawing templates with useful viewports, along with standard styles and settings for plan production objects (view frame groups, view frames, and match lines).

Setting up templates helps you work efficiently and ensures that final drawings conform to office standards.

When you use the plan production tools, you can use predefined templates that are specifically configured for use with plan production features. These templates are located in the Templates folder.

You can also use your own custom templates or existing AutoCAD Civil templates. If you choose to use templates other than the ones located in the Plan Production folder, an important prerequisite is that you first set the viewport types for plan view or profile view. This is explained in detail in Configuring Viewports for Plan Production (page 1700).

NOTE Only templates that contain viewports that have been specially configured with a Viewport Type of either "Plan" or "Profile" can be used with the plan production tools.

In addition to setting the viewport type, you may also want to adjust the size, scale, and other viewport settings for the templates you will use for plan production.

Following is a summary of the tasks you may need to perform on your templates before using the plan production tools:

■ Ensure that the Viewport Type property is set to either Plan or Profile.

■ Adjust viewport settings, such as size, scale, and so on.

■ Insert a north arrow block in the plan view viewport in the template (paper space).

NOTE When AutoCAD Civil is installed, templates that come with the product are installed to the following location by default: *C:\Documents and Settings \ <User Name >\Local Settings\Application Data\Autodesk\Civil2010\enu\Template.* The templates that are preconfigured with viewport types for plan view and profile view are located in: *C:\Documents and Settings \ <user name >\Local Settings\Application Data\Autodesk\Civil2010\enu\Template\Plan Production.*

**To prepare drawing templates for using the plan production tools**

1 Open a new drawing or template.

2 Select the default settings for the template.

For example, you may want to set the default object styles and label styles for all objects, and in particular for the new plan production objects (view frame groups, view frames, and match lines).

| For information about ... | See this Help topic |
| --- | --- |
| General AutoCAD Civil object styles | Object Styles (page 53) |
| General AutoCAD Civil label styles | Defining Label Settings (page 1475) |
| Plan production tool object styles and labels | Editing Plan Production Tool Settings (page 1701) |

**3** Configure viewport settings.

You'll want to configure your viewports as desired and capture this in the template(s). For example, you may want to adjust the size, scale, rotation, or other viewport settings. Also, you must make sure the Viewport Type for each viewport is set to Plan or Profile. If you use the AutoCAD Civil plan production templates provided in the Templates\Plan Production folder, the Viewport Types are already configured. However, if you use your own custom templates, or templates from previous AutoCAD Civil versions, you must set this property on the viewports. See Configuring Viewports for Plan Production (page 1700) for more information.

**4** Optionally, you may want to insert a north arrow block in the desired location in the layout.

**5** It is recommended that you lock the viewport(s).

**6** Save the drawing as a template (*.dwt*).

# Configuring Viewports for Plan Production

Before using custom templates for plan production, you must set the viewport type to either Plan or Profile.

By default, most of the templates provided with AutoCAD Civil have the Viewport Type property set to Undefined. However, the plan production templates that are located in the Template folder have viewports that are already configured to the appropriate viewport type: plan or profile.

The plan production templates that contain preconfigured Viewport Types are named:

*Civil 3D (Imperial) Plan and Profile.dwt*

*Civil 3D (Imperial) Plan Only.dwt*

*Civil 3D (Imperial) Profile Only.dwt*

If you want to use custom templates with the plan production features, you must set the Viewport Type to Plan or Profile, depending on the type of viewport you want.

If you use the predefined Plan Production templates located in the Template\Plan Production folder, the Viewport Types are already appropriately set.

While you are using the Create View Frames wizard, if you try to select a template that does not have appropriately defined viewports for the sheet type you select in the wizard (plan only, profile only, or plan and profile), AutoCAD Civil detects this and displays a message indicating that layouts containing the necessary viewports for the sheet type were not found.

**NOTE** When AutoCAD Civil is installed, templates that come with the product are installed to the following location by default: *C:\Documents and Settings \ <User Name >\Local Settings\Application Data\Autodesk\Civil2010\enu\Template\Plan Production\*.

**To set the Viewport Type for use with the plan production features**

**1** Open a drawing template.

**2** Make sure you are on a layout tab in paper space.

**3** Select the viewport that you want to display *plan* view data, right-click on it and then click Properties. The Properties palette is displayed.

> **NOTE** You can also display the Properties palette using one of the following methods:
>
> ■ Click the ⊞ Properties button on the Home tab of the ribbon.
>
> ■ Press Ctrl + 1.
>
> ■ At the command line, enter Properties.

**4** Scroll down to the Viewport section at the bottom of the Properties palette. Set the Viewport Type to Plan.

**5** Now right-click the viewport that you want to display *profile* view data, and set the Viewport Type to Profile.

**6** Save the template as *.dwt*.

# Editing Plan Production Tool Settings

Use view frame group settings to specify the default behavior for plan production tool commands.

Settings are handled in a standard way throughout AutoCAD Civil. You access settings using the Toolspace Settings tree. You can control settings at three levels: the drawing level, the object collection (feature) level, and the command level. For more information, see Understanding Settings (page 61).

Use the Settings tree View Frame Group collection shortcut menu to establish defaults for all plan production tool commands. You can change plan production tool-specific settings at this level, or override the drawing ambient settings.

Use the Commands collection in the View Frame Group collection to change plan production tool settings for a specific command.

> **NOTE** Overrides to the drawing ambient settings at the View Frame Group collection level and the Commands collection level affect only the specified level. The drawing level settings are not changed.

The topics in this section describe only those settings that affect plan production tool-related commands. They do not apply to the drawing ambient settings that you can change at the View Frame Group collection level and the View Frame Group command level, even though those settings are displayed in the Edit Feature Settings - View Frame Group dialog box. For more information about the drawing ambient settings, see Specifying Ambient Settings (page 68).

# Changing Plan Production Tool Creation Settings

Use the plan production tool creation and naming settings in the View Frame Group Settings (page 2081) dialog box to change plan production tool settings before you create view frames or sheets.

For example, before using the Create View Frames wizard, you should specify the default choices you want to use for plan production object names (object name format) and label styles. This will set your defaults for plan production objects (view frame groups, view frames, and match lines).

If a closed lock appears in the Lock column for a property, the property is locked at a higher level of the Settings tree. It cannot be changed at this level.

If you change a property value, a check mark is placed in the Override column. This override is also noted in the property table for related objects higher up the Settings tree, where an arrow is displayed in the Child Override column.

**To change plan production tool creation settings**

1 Do one of the following:

- **To edit settings for all plan production tool commands**: In Toolspace, on the Settings tab, right-click the View Frame Group collection. Click Edit Feature Settings. Do the same for the View Frame and Match Line collections on the Settings tab.

- **To edit settings for a specific command**: In Toolspace, on the Settings tab, expand the View Frame Group collection. Expand the Commands collection. Right-click the name of a command and click Edit Command Settings. For example, right-click the CreateSheets command and click Edit Command Settings.

The Edit Feature Settings (page 2081) dialog box is displayed.

2 To specify the default sheet creation and view frame creation options, right-click either the CreateSheets or CreateViewFrames commands and click Edit Command Settings.

3 To edit the default naming format for plan production tool objects (such as view frame groups, view frames, and match lines), expand the Default Name Format property group. Edit the Name Template settings for each item.

4 Click Apply.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Right-click View Frame Group collection ➤ Edit Feature Settings
OR
Settings tab: Right-click View Frame collection ➤ Edit Feature Settings
OR
Settings tab: Right-click Match Line collection ➤ Edit Feature Settings
OR
Settings tab: View Frame Group ➤ Commands ➤ Right-click <command-name> ➤ Edit Command Settings

Dialog Box

Edit Feature Settings (View Frame Group) (page 2081)
Edit Feature Settings (View Frame) (page 2084)
Edit Feature Settings (Match Line) (page 2084)

# Changing Plan Production Tools Default Style Settings

Use the view frame group settings to specify default styles and labels for view frame and match line creation.

The default styles specified in these settings are used to establish the default settings for the styles in the Create View Frame wizard, the View Frame Group Properties dialog box, the View Frame Properties dialog box, and the Match Line Properties dialog box.

**To change the plan production tool objects style and label settings**

1  In Toolspace, on the Settings tab, right-click the View Frame Groups collection. Click Edit Feature Settings to open the View Frame Group Settings (page 2081) dialog box.

2  Expand the Default Styles property group.

3  To specify default styles for plan production tool objects (view frames and match lines), click the corresponding field. Click [...] to open the Select Style dialog box.

4  To specify a default view frame or match line style, click View Frame Style or Match Line Style. Click [...] to open the Select Style dialog box.

5  To specify a default style for view frame or match line labels, click the corresponding field. Click [...] to open the Select Label Style dialog box.

6  To specify a default location for view frame or match line labels, click the corresponding field and select a location in the drop-down list.

7  Click Apply.

## Quick Reference

Toolspace Shortcut Menu

  Settings tab: right-click View Frame Group collection  ➤  Edit Feature Settings

Dialog Box

  View Frame Group Settings (page 2081)

# Creating View Frames

Use the Create View Frames wizard to quickly create view frames along an alignment.

You create view frames automatically by using the Create View Frames wizard. Start the wizard in any of the following ways:

■  Click Output tab ➤ Plan Production panel ➤ Create View Frames  .

■  In the Prospector tree, right-click View Frame Groups  ➤  Create View Frames.

■  At the command line, enter CreateViewFrames.

The view frames that are created represent rectangular areas along the alignment that will be displayed on plan/profile or plan-only sheets.

Before you create view frames, the desired alignment must already exist in your drawing. Depending on the type of sheets you want to produce (plan and profile or profile only), you may also need to have a profile already created. If you are creating a plan only view frame (or sheet set), then you do not need to have a profile in the drawing. For information, see Preparing for Plan Production (page 1699).

You can also insert a new view frame into an existing view frame group. For information, see Inserting View Frames (page 1714).

**To create view frames**

1  Do one of the following:

   ■ Click Output tab ➤ Plan Production panel ➤ Create View Frames 📇 .

   ■ In the Toolspace Prospector tree, right-click the View Frame Groups collection and click Create View Frames.

   The Create View Frames wizard is displayed.

   ---
   **NOTE** On any page of this wizard, you can click Create View Frames to create the view frames using the default choices on the wizard pages. If a criteria is needed that has not been supplied, then the Create View Frames button is unavailable (grayed out). You can also click the links on the left side of the wizard to go directly to a wizard page.

   ---

2  In the Alignments page (page 2084), select an alignment.

3  In the Station Range section, choose one of the following:
   ■ Automatic: Selects the entire alignment. So the start station is the start station of the alignment, and the end station is the end station of the alignment.

   ■ User Specified: When this is selected, you may enter a value or click 🖱 to specify a start and end location (stations) along the alignment in the drawing area.

4  Click Next to display the Sheets page.

**5** In the Sheets page (page 2085), in the Sheet Settings section, under Choose The Sheet Type You Want To Generate, select one of the following:

| Option | Result (Conceptual Image) |
|---|---|
| Plan And Profile | <br><br>The sheets that will be created will contain plan views and profile views. |
| Plan Only | <br><br>The sheets that will be created will contain plan views only (no profile views). |
| Profile Only | <br><br>The sheets that will be created will contain profile views only (no plan views). |

In the Template For Plan And Profile Sheet field, click ⁝⁝⁝ to open the Select Layout As Sheet Template dialog box and browse to the template to use for the sheets that will be created. The template you select must have viewports defined as plan and/or profile, depending on the sheet type selected in this dialog box (Plan And Profile, Plan Only, or Profile Only).

**6** Click OK in the Select Layout As Sheet Template dialog box.

**NOTE** If you select a template that does not have appropriately defined viewports for the sheet type you select (Plan Only, Profile Only, or Plan and Profile), AutoCAD Civil detects this and displays a warning message. For more information see, Configuring Viewports for Plan Production (page 1700).

**7** In the View Frame Placement section, select one of the following or accept the default:

| Option | Result (Conceptual Image) |
|---|---|
| Along Alignment | \nThe view frames will be aligned along the alignment. Typically, this is the preferred method for making the most efficient use of paper when plotting. |
| Rotate To North | \nThe view frames will be rotated according to the north arrow orientation of the drawing. |

**8** Optionally, you may select the check box next to the Set The First View Frame Before The Start Of The Alignment option, and then enter a value in the value field.

This sets the distance before the start station of the alignment that the first view frame is placed. Entering a distance here provides the specified amount of space before the selected alignment start so that the alignment start does not coincide with the edge of the viewport. If this check box is not selected, then this field is ignored, and the first view frame may be placed at the start of the alignment. Often the view frame start will be as desired without altering this option.

**9** Click Next to display the View Frame Group page.

**10** On the View Frame Group page (page 2086), specify a name, and optionally, a description of the view frame group object that will be created using this wizard.

**11** On the View Frame section, specify a layer, name, style, and label style for the view frames.

**12** In the Label Location field, choose the desired location to place the view frame labels. For example, choosing Top Left places the view frame labels at the top left side of the view frame.

**13** Click Next to display the Match Lines page.

**14** On the Match Lines page (page 2087), if you chose Plan Only, you can choose to insert match lines on the view frames by selecting the check box next to Insert Match Lines. Match lines are only displayed in model space and in plan views. They are not displayed in profile views.

---

**NOTE** If you do not want match lines included on the plan only view frames, clear the check mark from this option. This disables the options on this page. If you selected to create plan and profile or profile only sheets, this check box is automatically selected, and you cannot edit it (not available, grayed out).

---

**15** In the Positioning section, you may select one or both of the following options, and enter desired values in the text fields. Both these options let you adjust where match lines are placed, and how they may be moved after creation, so that they do not obstruct important drawing data.

- Snap Station Value Down To The Nearest: When this check box is not selected (cleared), match line positioning on the view frame is determined by using a rounding value based on derived stations. When this check box is selected, the match lines are positioned on the view frames based on the rounding calculation value entered. The rounding calculation always rounds down. For example, if the calculated station for a match line is 48+37.69, then a rounding of 100 would place the match line on 48+00.

| When Snap Station Value Down To The Nearest is set to ... | the station will round to |
|---|---|
| 100 | 18+00 |
| 50 | 18+50 |
| 10 | 18+60 |
| 1 | 18+65 |
| 27+45.3 to 50 | 27+00 to the next lowest multiple of 50 |

  **NOTE** This option will not accept values that cause the match lines to be placed in undesirable locations, such as before the previous match line or before the beginning of the alignment. If a rounding calculation would result in the match line being placed in an undesirable location, then the rounding calculation is ignored and the match lines are placed at calculated station.

- Allow Additional Distance For Repositioning (Increases View Overlap): Because the location of match lines is calculated automatically, it is possible that after you might need to move the match line slightly. For example, in some cases match lines or match line labels may be obscuring data of interest. You can use this option to increase the margin by which you can move (reposition) match lines. Using this option will have a side-effect of increasing the overlap area of view frames. When this check box is selected, you can move the match lines in plan view by the distance specified with this option. This ensures that the match line remains within (between) the intersection of the two neighboring view frames. You should enable this option if you want to be able to move the match lines by a certain distance. Note that having this option enabled is useful if you need to force an overlap of view frames when, on a straight line alignment, adjacent edges or sides of two view frames are at the same location, thereby not allowing enough room for labels.

**16** In the Match Line section, specify a layer, name, and style for the match lines that will be created.

**17** In the Labels section, specify the labeling criteria for the match line labels, including location.

**18** Click Next to display the Profile Views page.

  **NOTE** The Profile Views page is not available (grayed out) if a sheet type of Plan Only was chosen on the Sheet Set page of this wizard.

**19** On the Profile Views page (page 2089), verify that an appropriate profile view style and band set have been selected. If desired, you may select a different profile view style and band set.

  **NOTE** It is important to understand that the profile view style and band set for the profile view cannot be changed later when you are using the Create Sheets wizard to create sheets. For more information, see Understanding Profile View Options (page 1724).

**20**   Click Create View Frames to create the view frames.

When the wizard has completed creating the view frames, you will notice the following:

- The view frames and match lines are created in the drawing.

- In the Prospector tree, the following objects are created: one view frame group, one or more view frames in a view frames collection, one or more match lines in a match lines collection.

**NOTE** If the Insert Match Lines check box on the Match Lines page was not selected (cleared), then match lines are not created.

Now that the view frames are created, you may want to adjust the location or labels of the view frames and match lines. See Moving View Frames (page 1711) or Moving Match Lines (page 1716) for more information.

After you are satisfied with how your view frames and match lines are positioned, the next step is to create the sheets using the Create Sheets wizard. See Creating Sheets (page 1719) for step-by-step information.

### Quick Reference

Ribbon

  Output tab ➤ Plan Production panel ➤ Create View Frames

Menu

  General menu ➤ Plan Production Tools ➤ Create View Frames

Toolspace Right-Click Menu

  Prospector tab: right-click View Frame Groups collection ➤ Create View Frames

Command Line

  CreateViewFrames

Dialog Box

  Create View Frames Wizard (page 2084)

# Using View Frame Groups

Create view frame groups using the Create View Frames wizard.

After view frame groups are created, you can edit certain properties of the view frame group, or delete it. Although you can move view frames and match lines within a view frame group, view frame groups themselves are not designed to be moved as an entity.

You can delete individual view frames within a view frame group. You can also delete the entire view frame group. Deleting the entire view frame group object automatically deletes all the view frames and match lines associated with it.

**NOTE** View frame groups are not dynamically associated with the sheet files that are created from them. Updating a view frame group, moving match lines, or moving view frames will not affect the sheet files. The same is true with the sheet files created from the view frame groups. Updating the sheet files has no impact on the view frame groups.

After a view frame group is created, you can delete the alignment from which it was created, if you need to. Labeling functionality, such as the ability to display the sheet number in labels in the view frames, is still maintained even if the original alignment from which the view frame group was created is deleted.

# Creating View Frame Groups

Use the Create View Frames wizard to create view frame groups.

On the third page of the Create View Frames wizard—the View Frame Group page—you specify the object creation criteria for the view frame group object. For example, you enter a name for the view frame group object, and you can select the layer for the view frames. For more information see, View Frame Group page (page 2086) of the Create View Frames wizard.

Note that you can also create a reference to an existing view frame group data shortcut. For more information, see Creating a View Frame Group Reference (page 1709).

You can also insert a new view frame in to an existing view frame group. See Inserting View Frames (page 1714) for more information.

# Creating a View Frame Group Reference

You can create a reference to an existing view frame group in a data shortcut (page 2472) or a Vault project.

The view frame group reference is a lightweight read-only copy of the original. By means of data references, you can reuse a view frame group among several drawings to create consistency in the plan production sheets. Before you create the view frame group reference, it must exist in the current project collection on the Toolspace Prospector tab.

Before you create a view frame group reference, you must first create a reference to the related alignment.

**See also:**

■  Creating Data Shortcuts (page 131)

■  Vault: Checking a Drawing in to a Project (page 158)

**To create a reference to a data shortcut view frame group**

1  In Toolspace, on the Prospector tab, ensure that the correct project folder is identified on the Data Shortcuts node.

2  Expand the Data Shortcuts ➤ View Frame Groups collection, right-click the desired group, and then click Create Reference.

   The Create View Frame Groups Reference dialog box is displayed, in which you can optionally change properties of the reference view frame group, as described in the following steps.

3  In the Create View Frame Group Reference dialog box, to change the source of the reference, select the source in the Source View Frame Group list.

4  Enter a unique name for the view frame group and optionally, a description.

5  For the view frames and match lines included in this view frame group, you can specify a style, label style, and layer, or accept the defaults.

6  Click OK to create the view frame group reference.

   In the Prospector tree, the view frame group reference is displayed under the View Frame Groups collection with a  next to it.

**To create a reference to a view frame group in a Vault project**

1  In Toolspace, on the Prospector tab, in Master View, expand the Projects ➤ <project name> ➤ View Frame Groups collection, right-click the desired view frame group, and then click Create Reference.

The Create Surface Reference dialog box is displayed, in which you can optionally change properties of the reference group, as described in the following steps.

2   In the Create View Frame Group Reference dialog box, to change the source of the reference, select the source in the Source View Frame Group list.

3   Enter a unique name for the view frame group and optionally, a description.

4   For the view frames and match lines included in this view frame group, you can specify a style, label style, and layer, or accept the defaults.

5   Click OK to create the view frame group reference.

In the Prospector tree, the view frame group reference is displayed under the View Frame Groups collection with a  next to it.

### Quick Reference

Toolspace Right-Click Menu (for data shortcut project)

Prospector tab: Data Shortcuts ➤ View Frame Groups ➤ <view frame group name> ➤ Create Reference

Toolspace Right-Click Menu (for Vault project)

Prospector tab: Projects ➤ <project name> ➤ View Frame Groups ➤ <view frame group name> ➤ Create Reference

Command Line

CreateViewFrameGroupReference (for use only with view frame groups in Vault)

Dialog Box

Create View Frame Group Reference (page 2203)

## Editing View Frame Groups

You can edit certain properties of a view frame group, or add a new view frame to a view frame group.

For example, you can change a view frame group's object name, description, or the style of the view frames and match lines, as well as the layers on which they are displayed. For more information, see the View Frame Group Properties dialog box (page 2095).

For information on adding a view frame to a view frame group, see Inserting View Frames (page 1714).

The following restrictions apply to editing view frame groups:

■   Although you can move view frames and match lines within a view frame group, view frame groups themselves are not designed to be moved as an entity.

■   You can delete an entire view frame group, which deletes all view frames, match lines, and labels associated with the group.

■   You can delete individual view frames or match lines within a view frame group.

## Deleting View Frame Groups

Deleting a view frame group also deletes all view frames, match lines, and labels associated with that view frame group.

Any sheets or sheet sets that have already been created from that view frame group are not affected.

If you explode one or more view frames or match lines, all view frames and match lines in that view frame group are exploded, and the following objects are deleted: view frames, match lines, and the view frame group.

**To delete a view frame group**

1 Click a view frame in the group. Click Plan Production tab ➤ Modify View Frame panel ➤ ▼ ➤ Delete Group ▣ .

2 The view frame group, and all view frame and match line objects belonging to it, are removed from the View Frame Groups collection on the Prospector tab. The view frames and match lines are automatically removed from the drawing.

## Quick Reference

Ribbon

Click a view frame in the group. Click Plan Production tab ➤ Modify View Frame panel ➤ ▼ ➤ Delete Group ▣

Toolspace Right-Click Menu

Prospector tree: Right-click View Frame Groups collection ➤ View frame group ➤ Delete

Command Line

DeleteViewFrameGroup

# Editing View Frames

You may want to edit view frame styles, view frame label styles, or edit, move or delete view frames or their labels.

Like other AutoCAD Civil objects, you can edit certain properties of a view frame. For example, you can change a view frame's object name, description, object style, or the layer on which it is displayed.

View frame objects are displayed in the Prospector tree, and you can control their default style and labeling from the View Frame collection in the Settings tree. For more information, see the View Frame Properties dialog box (page 2093).

View frames can include a label that displays the information you want. For example, you may want your view frame labels to identify the area along the alignment it is displaying, or reference more specific data from that area.

You can move or rotate a view frame, but you cannot resize it. You can also insert or delete view frames within a view frame group.

# Moving View Frames

After view frames are created, they can be moved or rotated using grips.

The center, slider, and rotation grips on the view frame enable you to move or rotate it.

Although you can move or rotate view frames, you cannot change the size of a view frame. This is because the view frame's size is based on the size of the viewport it references in the associated template.

You will notice that when you move a view frame, the view frame stays attached to the alignment it is associated with. If a view frame is moved off the alignment, and you want to reattach it to the alignment,

be certain that OSNAPS are turned on, and you can drag the center grip towards the alignment. When you move a view frame that has a view frame label on it, the label moves with the view frame.

Moving a view frame does not move any match lines associated with that view frame. When moving a view frame, the match lines associated with the view frame stay positioned where they intersect the alignment, and do not move.

If you need to move view frames *after* sheets have been created, instead of moving the view frames, you should make these changes to the sheet by editing the viewport. If you move view frames after sheets have been created, these changes will not be reflected in any sheets that have already been created from the view frames. In order to see those changes reflected in the sheets, you will have to recreate the sheets.

If the alignment associated with a view frame has been deleted, the view frame remains, but the slider grip is no longer displayed, and the view frame can be moved in any direction.

Also, if the view frame is set to align with a north arrow block in the drawing, the rotation grip is not displayed.

For more information about grips, see "Using Grips to Edit Objects" in the AutoCAD Help.

**To move or rotate view frames using grips**

1  In either model space, click the view frame you want to move or rotate.

2  Click one of the following grips on the view frame:

   ■  ▢ Center grip: Moves the view frame freely in any direction.

   ■  ◇ Slider grip: Moves the view frame forward or backward along the alignment. If the view frame is moved off the alignment, the offset to the alignment is maintained. If the view frame remains parallel to the tangent to the alignment, the grips stay on alignment.

   ■  ● Rotation grip: Rotates the view frame about the center of the view frame.

   **NOTE** If the view frame is set to align with a north arrow block in the drawing, the rotation grip is not displayed (not available).

3  Drag the grip to move or rotate the view frame.

# Deleting View Frames

You can delete a view frames in a view frame group, or delete the entire view frame group.

You can delete a view frame in a view frame group. Or you can delete all view frames within a view frame group by deleting the entire view frame group in Prospector. Deleting a view frame group deletes all view frames and match lines within a view frame group, in addition to deleting the view frame group object as well.

When you delete a view frame group, any sheets and/or sheet sets that may have already been created by the view frame group are not affected.

If you explode one or more view frames or match lines, all view frames and match lines in that view frame group are exploded, and the following objects are deleted: view frames, match lines, view frame group.

**NOTE** When two view frames share a common match line, and you delete one of the view frames, the shared match line is not removed.

For more information, see Deleting View Frame Groups (page 1710).

**To delete a single view frame**

■ In the Prospector tree, expand the desired View Frame Group object, then right-click a view frame and click Delete.
The view frame and its associated match line object(s) are removed from the View Frames and Match Lines collections in the Prospector tree, and in the drawing.

**NOTE** When two view frames share a common match line, and you delete one of the view frames, the shared match line is not removed.

# Labeling View Frames

You can include or omit labels on view frames, specify their location, and you can design and specify view frame label styles that suit your needs.

Use the View Frame Group page in the Create View Frames wizard to select a label style and label location for view frames in a view frame group.

If you do not want labels on a view frame, you can turn off the visibility of the labels in the label style.

Use the Label Location options on the View Frame Group page in the Create View Frames wizard to select where the view frame labels will be placed. For example, selecting Top Left places the view frame label at the top left corner of the view frame.

You can delete view frame labels. Selecting any view frame label automatically selects all view frame labels associated with that view frame group. You can delete all view frame labels in a view frame group by selecting them and pressing the Delete key. To delete individual labels, Ctrl-click on a label to select it and press Delete.

In the View Frame collection in the Toolspace Settings tree, you can create and edit view frame label styles. See Creating and Editing Label Styles (page 1477) for more information.

**To edit view frame label styles**

1 In the Settings tree, expand the View Frame collection, and then expand the Label Styles collection.

2 Double-click the Standard label style.

3 Use the Label Style Composer to edit the default Standard view frame label style, or create your own custom label style for view frames.

**To edit existing view frame labels**

1 In the drawing, do one of the following:

■ to select all view frame labels in a view frame group, click on any view frame label

■ to select a single view frame label, use Ctrl + left-click on a view frame label

2 Right-click and select Edit Labels on the right-click shortcut menu.

3 Use the Edit View Frame Labels dialog box to select a different label style, create a new label style, edit the existing label style, or change the label location.

**To reset view frame labels**

1 In the drawing, do one of the following:

■ to select all view frame labels in a view frame group, click on any view frame label

■ to select a single view frame label, use Ctrl + left-click on a view frame label

**2** Right-click and select Reset Labels on the right-click shortcut menu.

### Quick Reference

Toolspace Shortcut Menu

Settings tab: View Frame ➤ Label Styles

Dialog Box

Label Style Composer Dialog Box (page 1942)

# Inserting View Frames

You can insert a new view frame into a view frame group.

In most cases, the new view frame is inserted before the selected view frame.



You can insert a new view frame by

■ right-clicking on a view frame and clicking Insert View Frame

■ entering the InsertViewFrame command and specifying an insertion point in the drawing

Each of these methods is described below.

### Right-clicking on a View Frame and Clicking Insert View Frame

When you insert a new view frame by right-clicking on an existing view frame and clicking Insert View Frame, or if you use the InsertViewFrame command and select an existing view frame, the new view frame is inserted as follows:

■ If there is no gap (blank space) between the selected view frame and its previous view frame, the new view frame is inserted before the selected view frame (towards the beginning of the alignment).

■ If there is a gap (blank space) between the selected view frame and its previous view frame, the new view frame is inserted by finding the center (at the alignment) of the two match lines, and inserting the new view frame centered on this point. If the selected view frame is the first view frame in the view frame group, and therefore, has no left match line, a new left match line is created.

**Specifying an Insertion Point in Drawing**

When you insert a new view frame by using the InsertViewFrame command and you enter S to specify an insertion location, the following rules are used:

■ If the selected point is a valid location for a view frame, the new view frame is inserted, centered in the selected point.

■ If the selected point is not a valid location for a view frame, a message indicates that the location is not valid for view frame insertion, and the new view frame is not inserted.

A valid location is some other point along the alignment, or close to the alignment.

**To insert a view frame into a view frame group**

➤ Do one of the following:

■ In the drawing area, right-click on a view frame and click  Insert View Frame.

■ At the command line, enter InsertViewFrame, and then click on a view frame, or select an insertion point in the drawing area.

The new view frame is inserted.

## Quick Reference

Drawing Area Shortcut Menu

Right-click <view frame object> ➤ Insert View Frame

Command Line

InsertViewFrame

# Editing Match Lines

You can edit match line styles, match line label styles, or edit, move or hide match lines or their labels.

Like other AutoCAD Civil objects, you can edit certain properties of a match line. For example, you can change a match line's object name, description, object style, or the layer on which it is displayed. Match line objects are displayed in the Prospector tree, and you can control their default style and labeling in the Match Line collection in the Settings tree. For more information, see the Match Line Properties dialog box (page 2094).

Match lines can include a label containing information that you want. For example, you may want your match line labels to identify the previous and next sheet (view frame).

You can edit a match line length using grips, or move match lines. Use the options on the Match Lines page of the Create View Frames wizard to control how match lines are positioned on alignments, and by how much distance match lines can be repositioned (moved) after you create them.

The only way match lines can be deleted is to delete the view frame, or the view frame group, it belongs to. Use the match line style to hide the visibility of match lines if you do not want them to display. For more information, see Deleting Match Lines (page 1717).

# Editing the Match Line Mask Area

The Match Line Style (page 2099) dialog box contains options for setting a hatch pattern in the mask area.

The mask is displayed in paper space in the area between the outside of a match line and the inside boundary of the view frame, indicated by the dark shaded areas in the following illustration:



The match line mask area is the area that exists between the outside of a match line and the inside border or boundary of the view frame. On the Display tab of the Match Line Style dialog box, there is an option to set the hatch pattern for this match line mask area.

Match line labels are displayed on top of the match line mask area, and are therefore visible over it.

**To edit the match line mask area**

1   In the Settings tree, expand the Match Line collection, and then expand the Match Line Style collection.

2   Double-click a match line style to open the Match Line Style dialog box.

3   Click the Display tab.

4   Select a View Direction (2D or 3D). You can set different hatch patterns for 2D and 3D.

5   In the Component Hatch Display section at the bottom, click the current value for Pattern. The Hatch Pattern dialog box is displayed.

6   Select a hatch type from the list. Optionally, you may browse to another hatch pattern palette. For more information about hatch patterns, see "Overview of Hatch Patterns and Fill" in the AutoCAD Help.

7   Click OK in the Hatch Pattern dialog box.

8   Optionally, you may specify Angle and/or Scale.

9   Click either Apply or OK on the Display tab of the Match Line Style dialog box.

# Moving Match Lines

You can move match lines in plan view if you have enabled the match line repositioning option during view frame creation.

The following match line positioning options are displayed on the Match Lines page in the Create View Frames wizard:

■   Snap Station Value Down To The Nearest

■   Allow Additional Distance For Repositioning (Increases View Overlap)

**Snap Station Value Down to the Nearest**

When the Snap Station Value Down To The Nearest check box is selected, match line positioning is determined by using the value entered in the Distance field, which is in drawing units.

The rounding calculation always rounds down. The following table lists examples of this calculation based on the value entered.

| When Snap Station Value Down To The Nearest is set to ... | The station rounds to |
| --- | --- |
| 100 | 18+00 |
| 50 | 18+50 |
| 10 | 18+60 |
| 1 | 18+65 |
| 27+45.3 to 50 | 27+00 to the next lowest multiple of 50 |

**NOTE** This option will not accept values that cause the match lines to be placed in undesirable locations, such as before the previous match line or before the beginning of the alignment. If a rounding calculation would result in the match line being placed in an undesirable location, then the rounding calculation is ignored and the match lines are placed at calculated station.

When this check box is not selected (cleared), match line positioning on the view frame is determined by using a rounding value based on derived stations.

**Allow Additional Distance For Repositioning (Increases View Overlap)**

When this check box is selected, you can move match lines within a specified limit. Enabling this option can also be used to force an overlap of frames in plan sheets only when, on a straight line alignment for example, adjacent edges of two view frames might be coincident, and therefore not provide enough room on either side of the match line for labeling and notes.

Note that the value entered here is the total distance that is removed from the calculated distance between match lines. It will allow for match lines to be moved apart to the total calculated distance. However, AutoCAD Civil will not allow you to enter a distance that is greater than the view frame width.

**To move match lines**

1 In either model space or paper space, click the match line you want to move.

2 Click one of the following grips on the match line:

- **Center grip**: Moves the match line freely in any direction.

- **Slider grip**: Moves the match line forward or backward along the alignment.

- **Rotation grip**: Rotates the match line about the center of the match line.

3 Drag the grip to move or rotate the view frame.

# Deleting Match Lines

Match lines are deleted automatically when you delete a view frame or view frame group.

After match lines are created, the only way to delete them is to delete the view frame, or view frame group object they belong to. For more information, see Deleting View Frames (page 1712) or Deleting View Frame Groups (page 1710).

If you explode one or more match lines or view frames, all match lines and view frames in that view frame group are exploded, and the following objects are deleted: match lines, view frames, view frame group.

You can delete match line labels. Selecting any match line label automatically selects all match line labels in that view frame group. To delete them individually, select them by using Ctrl + click, and then press the Delete key.

If you do not want match lines to display, change the display style so that the match line component's visibility is turned off. For more information, see Match Lines Styles and Display (page 1697).

## Labeling Match Lines

You can insert or omit labels on match lines, specify label location, and design and specify match line label styles that suit your needs.

Use the Match Lines page of the Create View Frames wizard to select a label style and label location for the left and right side match lines.

If you do not want labels to be displayed on match lines, turn off their visibility using the match line label style.

Use the Left Label Location and the Right Label Location options on the Match Lines page in the Create View Frames wizard to select where the match line labels will be placed. Select from the following locations:

- **Start**: Places the match line label at the start of the match line, near the top of the view frame.

- **End**: Places the match line label at the end of the match line, near the bottom of the view frame.

- **Middle**: Places the match line label at the middle of the match line.

- **Alignment Intersection**: Places the match line label where the alignment intersects the match line.

In the Match Line collection in the Toolspace Settings tree, you can create and edit match line label styles. For the match line text component, by default you can select from the following properties to add to a match line label:

- Match Line Name

- Match Line Number

- View Frame Group Alignment Name

- View Frame Group Name

- Match Line Station Value

- Match Line Raw Station Value

- Previous Sheet Number

- Next Sheet Number

**NOTE** If you use the Previous Sheet Number or the Next Sheet Number properties in your match line labels, this information displays as "####" in the match line label until the sheets are generated. After the sheets are generated, the match line label is populated with appropriate sheet number information.

See Creating and Editing Label Styles (page 1477) for more information.

**To edit match line label styles**

1 In the Settings tree, expand the Match Line collection, and then expand the Label Styles collection.

2 Expand the Match Line Left collection. Double-click the Standard label style.

3 Use the Label Style Composer to edit the default Standard match line label style, or create your own custom label style for the left side match line.

4 Repeat step 3 for the right side match line label.

**To edit existing match line labels**

1 In the drawing, do one of the following:

- to select all match line labels in a view frame group, click on any match line label

- to select a single match line label, press and hold the Ctrl key while clicking on a match line label

2 Right-click and select Edit Labels on the right-click shortcut menu.

3 Do one of the following:

- use the Edit Match Line Group Labels dialog box to edit labels for all match lines in the view frame group

- use the Edit Match Line Label dialog box to edit a single match line label

You can select a different label style, create a new label style, edit the existing label style, or change the label location for the selected match lines.

**To reset match line labels**

1 In the drawing, do one of the following:

- to select all match line labels in a view frame group, click on any match line label

- to select a single match line label, press and hold the Ctrl key while clicking on a match line label

2 Right-click and select Reset Labels on the right-click shortcut menu.

## Quick Reference

Toolspace Shortcut Menu

Settings tab: Match Line ➤ Label Styles ➤ Match Line Left
Settings tab: Match Line ➤ Label Styles ➤ Match Line Right

Dialog Box

Label Style Composer Dialog Box (page 1942)

# Creating Sheets

Use the Create Sheets wizard to quickly create sheets for construction documents (plans).

After you have used the Create View Frames wizard to create view frames, you can create sheets automatically using the Create Sheets wizard.

You can specify whether the sheets are created in the current drawing or in one or more new drawings.

**NOTE** If you choose to create sheets that display profile views, the profile views will be created and placed in the drawing automatically in model space in the file containing the sheets.

Before you can use the Create Sheets wizard, in the drawing you must have already created a view frame group using the Create View Frames wizard.

It is important to note that the profile view style and band set that were used during the Create View Frames wizard are used to calculate the placement of view frames. These styles cannot be changed later for the view frame group.

**To create sheets**

1 Do one of the following:

■ Click Output tab ➤ Plan Production panel ➤ Create Sheets 📑 .

■ In the Prospector tree, expand the View Frames Groups collection, right-click a View Frame Group ➤ Create Sheets.

■ At the command line, enter CreateSheets.

**NOTE** On any page of this wizard, you can click Create Sheets to create the sheets using the default choices on the wizard pages. If a criteria that is needed that has not been supplied, then the Create Sheets button is not available (grayed out). You can also click the links on the left side of the wizard to go directly to a wizard page.

2 On the View Frame Group And Layouts page (page 2089), select a view frame group from the list.

**NOTE** As a reminder, the sheet type that was selected for this view frame group is displayed beneath the View Frame Group list.

3 For View Frame Range, all view frames are selected for processing by default, but you can choose a single view frame, or multiple view frames from the group by clicking Selection and then clicking the Choose View Frames ... button.

4 In the Layout Creation section, choose how the layouts will be created. A conceptual image on the right side of the wizard page provides an indication of the result.

| Option | Description |
| --- | --- |
| Number Of Layouts Per New Drawing (1) | This option may be an appropriate choice if you plan to have several people working |

| Option | Description |
|---|---|
| | on individual sheets. When this option is selected, and you enter 1 for the value, one new layout (sheet) is created in each new drawing. The total number of sheets and drawings depends on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. |
| Number Of Layouts Per New Drawing (>1) | If you choose this option and enter a value greater than 1 (3, for example), then 3 new layouts are created in each new drawing. The total number of sheets and drawings depends on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. |
| All Layouts In One New Drawing | This option may be an appropriate choice if having each individual layout in a separate drawing is not necessary for you, and if you have less than 10 sheets. You may place all the layouts in one newly created drawing. The total number of sheets depends on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. For optimal results, it is recommended that you create no more than 10 sheets per drawing. Therefore, this option may not be the best choice if you have more than 10 sheets. |
| All Layouts In The Current Drawing | If you are generating less than 10 sheets, you may want to generate the layouts in the current drawing. The total number of sheets depends on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. |

**NOTE** For optimal results, it is recommended that you create no more than 10 sheets per drawing. Therefore, this option may not be the best choice if you have more than 10 sheets.

5 Specify a name for the layouts or accept the default naming template.

6 Optionally, you can use the Choose The North Arrow Block To Align In Layouts option to automatically align a north arrow block in the sheet(s). Any blocks that exist in the referenced template (*.dwt*) are

displayed in the drop-down list. If no blocks exist in the referenced template, the list displays None. If you select a block from this list, that block will be aligned to the north on the sheet(s).

**7** Click Next to display the Sheet Set page.

**8** On the Sheet Set page (page 2090), do one of the following:

■ If you want to manage the sheets using the Autodesk Vault project management features, and you are already logged in to the desired vault, the Add Files To Vault is automatically selected.

■ If you want to manage the sheets using the Autodesk Vault project management features, and you are not logged in to the desired vault, click Log In To Vault ..., log in to the desired project, and then return to the Sheet Set page and click Add Files To Vault.

■ If you do not want to manage the sheets using the Autodesk Vault project management features, make sure the Add Files To Vault check box is not selected (cleared).

**NOTE** If you choose to manage sheets using the Autodesk Vault project management features, the sheet set file (.*dst*), and the sheet files (.*dwg),* will be added to the vault project you selected. If Autodesk Vault is not currently installed on the machine you are using, these options are not available (grayed out and not available for edit).

**9** In the Sheet Set section, select either New Sheet Set or Add To Existing Sheet Set.

**10** In the Sheet Set File (.DST) Storage Location field, specify a location where the sheet set file (.*dst*) will be stored. If you selected Add To Existing Sheet Set, this field displays the path of the sheet set file for the sheet set selected in the Add To Existing Sheet Set field. If you want to use the Sheet Set Manager later to manage this sheet set, you will need to know this location and filename.

**11** In the Sheets section, specify a storage location and name for the sheet file(s) that will be created during this wizard session. If you have chosen to save all sheets in the current drawing (the All Layouts In The Current Drawing choice on the Create Sheets wizard View Frame Group and Layouts page), these fields are not available (read only, grayed out).

■ Sheet Files Storage Location: Specifies the location where the sheet files created during this session will be located. Click ⬚ to select a location. If you are logged in to a project in Vault, by default this path displays the root folder for the project. If you are not logged in to a project in Vault, by default this path is set to the directory containing the working (current) drawing.

■ Sheet File Name: Specifies the name of the sheet file(s) that will be created.

**12** Click Next to display the next page in the wizard sequence. If the view frames do not include any profile views, then the Profile Views page of the Create Sheets wizard is not available, and the Data References page is displayed next.

**13** On the Profile Views page (page 2091), you can specify additional settings for profile views. Note that the profile view style and the band set that were selected during view frame creation cannot be changed at this stage; however, the current profile view style and band set that will be used are displayed as read-only on this page. Only additional settings relating to profile views (Other Profile View Options) can be changed at this part of the process. For more information, see Understanding Profile View Options (page 1724).

**NOTE** If you need to change these profile view or band set style settings, you must cancel out of the Create Sheets wizard and recreate the view frames, using the updated style choices for the profile views and/or the band sets.

**14** In the Other Profile View Options section, you can get other settings from an existing profile view, or you can choose settings by launching the Profile View wizard at this point (from this page of the Create Sheets wizard). If you choose to launch the Profile View wizard from the Create Sheets wizard, you will

be returned to the Profile Views page of the Create Sheets wizard when you are finished selecting profile view options.

**15** If you are creating plan and profile sheets, you can choose how to align the data that is displayed in the plan and profile views using the Align Views options. If you are creating plan only or profile only sheets, these options are not available. These options are useful on projects that require the data displayed in sheets to be aligned according to certain project requirements (such as aligned left, center, or right).

- Align Profile And Plan View At Start: When this option is selected, the left side (start station) match line in the plan view aligns with the left side (start) of the profile view.

- Align Profile And Plan View At Center: When this option is selected, the center of the plan view aligns with the center point of the profile view.

- Align Profile And Plan View At End: When this option is selected, the right side (end station) match line in the plan view aligns with the right side (end) of the profile view.

**16** Click Next to display the Data References page.

> **NOTE** The Data References page is not available if you chose All Layouts In The Current Drawing in the Layout Creation section of the View Frame Group and Layout page.

**17** On the Data References page (page 2092), you can select or omit the data you want to be included in your sheets. Select the check box next to the data references you want displayed in the sheets. Some references cannot be removed from the selection, such as the alignment and profile associated with these sheets. If you are including pipe networks in the destination drawing(s), you may want to check the Copy Pipe Network Labels To Destination Drawings to also copy the labels associated with any selected pipe networks.

**18** Click Create Sheets to create the sheets.

**19** If you selected to create sheets containing profile views, you are prompted to select an insertion point in model space to create the profile views. Click a location in the drawing (model space) to insert the profile views.

You will only see profile views in the current drawing if you selected the All Layouts in the Current Drawing option in the Layout Creation section of the View Frame Group and Layout page.

Depending on your Event Manager settings, Event Manager messages may inform you of the location of the sheet sets, as well as the number of layouts created.

Once the sheets are created, the Sheet Set Manager automatically displays, showing the newly created sheets in a sheet set. For more information about the Sheet Set Manager, see the Sheet Set Manager Help topics in the AutoCAD Help.

## Quick Reference

Ribbon

Output tab ➤ Plan Production panel ➤ Create Sheets

Menu

General menu ➤ Plan Production Tools ➤ Create Sheets

Command Line

CreateSheets

Dialog Box

# Understanding Profile View Options

When using the plan production wizards, it is important to understand how profile view options affect the data displayed in profile views.

Sizing of the profile view in the viewport requires information from the profile view style and band set. Together these items determine the extents of the data that will be presented in the viewport. Because the profile view style and the band set determine the extent of the data presented in the view port, these items must be configured as desired before starting the Create View Frames wizard, and then selected at the time that the view frames are created (when using the Create View Frames wizard).

There are other profile view settings, however, that can be specified during the creation of the profile view, or rather, while using the Create Sheets wizard. These settings include split profile view options, profile display options, pipe network display options, and data band information.

Changing these settings is not required until the profile views are actually created. These profile view settings can be specified in one of two ways on the Profile Views page of the Create Sheets wizard. These two different methods of changing profile view information during view are described in the following table.

| Method of changing profile view information | Description |
| --- | --- |
| Selecting an existing profile view | Selecting an existing profile view **does not override** the profile view style and band set information used when the view frames were created. |
| Using the Create Multiple Profile View wizard | Using this option will allow you to step through the Create Multiple Profile View wizard with several values pre-populated (some of which cannot be changed, such as style, station range, etc.).<br>The options that will either be extracted from the selected profile view, or gathered from the Create Multiple Profile View wizard, and subsequently used in creating profile views for sheets include:<br>■ split profile view options<br>■ profile display options<br>■ pipe network display options<br>■ data band (band set) information |

If desired, you can select an existing profile view when you are using the Create View Frames wizard, and then, when you are using the Create Sheets wizard, you can modify that profile view using the Multiple Profile View wizard.

# Using Vault with Plan Production Tools

This section describes how to use Autodesk Vault with plan production tools.

You may want to manage the objects and files that are both consumed (used) by and created from the plan production tools with Autodesk Vault. If you choose to do so, you will be using an efficient mechanism for maintaining your organization's standards for plan set drawing templates.

If you choose to use Autodesk Vault with the plan production features, the following files associated with plan production will be added to the selected project in Autodesk Vault:

■ sheet set data file (*.dst*),

■ the current drawing (*.dwg*)

If you are saving sheets to the current drawing file, and the current drawing file is already in the vault, that drawing file cannot be checked in to vault automatically. This must be done by you.

The Add To Autodesk Vault check box is not available for editing (grayed out) when sheets are being saved to the current file. In other words, when All Layouts In The Current Drawing is selected on the View Frame Group And Layouts page in the Create Sheets wizard, the Add To Autodesk Vault option is also not available (grayed out).

# Using Data References with Plan Production Tools

You can use data references with plan production tools.

When using the Create Sheets wizard, you can select the objects in the drawing that you want to include or exclude from the resulting sheets.

To do this, use the Data References page in the Create Sheets wizard, where you can select the objects that you want to be referenced in the sheets. You can select the following object types:

■ surfaces

■ alignments

■ profiles

■ pipe networks

When you select an object by placing a check mark next to it, a data reference (data shortcut) to that object is created in the sheet. This provides a way for you to include or exclude certain objects from the sheets.

Including data references in your sheets instead the actual data ensures that the geometry of the source data can be modified only in the source drawing, not in the resulting sheet drawing. However, display styles and labels for the reference object can be modified in the sheet drawing.

Another benefit of using data references is that if the source data objects change, the information displayed in the sheets dynamically updates.

For more information, see Using Data Shortcuts (page 127).

# Plan Production Tools Command Reference

You can use commands to quickly access plan production tools.

The following table lists the plan production tool AutoCAD Civil commands and briefly describes their functionality.

| Command | Description |
| --- | --- |
| CreateSheets | Displays the Create Sheets wizard for creating sheets. (page 1719) |
| CreateViewFrames | Displays the Create View Frames wizard for creating view frames. (page 1703) |

| Command | Description |
| --- | --- |
| DeleteViewFrameGroup | Deletes the selected view frame group (page 1710) |
| EditMatchLineProperties | Specifies the match line style, name, and description (page 2094) |
| EditMatchLineStyle | Controls the display of match lines and the hatch pattern of the match line mask (page 1715) |
| EditViewFrameGroupProperties | Specifies view frame and match line styles, names, and descriptions for the entire view frame group (page 2095) |
| EditViewFrameLabels | Edits view frame label style and location (page 1713) |
| EditViewFrameProperties | Specifies the view frame name, description, and style (page 2093) |
| EditViewFrameStyle | Controls the view frame border visibility and color (page 1695) |
| InsertViewFrame | Inserts a new view frame into a view frame group (page 1714) |

# Command Reference

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AutoCAD Civil adds its own unique civil engineering and surveying commands to the AutoCAD command set.

This section provides information about the commands in AutoCAD Civil.

For information about AutoCAD commands, see the *AutoCAD Command Reference*.

## Accessing Commands

You can access many AutoCAD Civil commands using more than one method.

In Help, click the Quick Reference tab on a tabbed help topic to display a table providing the methods available for accessing a specific command. Not all methods are available for all commands. Where applicable, the following information is available on the Quick Reference tab:

| | |
|---|---|
| **Ribbon** | Describes how to access the command from the ribbon. |
| **Menu** | Describes how to access the command from the cascading menu. |
| **Toolspace Shortcut Menu** | Describes how to access the command from Toolspace by right-clicking an object in the Prospector or Settings tree. |
| **Object Shortcut Menu** | Describes how to access the command from a drawing by right-clicking the applicable object. |
| **Command Line** | Describes how to access the command from the command line. |
| **Toolbar** | Shows the toolbar icon to click or describes how to access the command from a toolbar. |
| **Dialog Box** | Provides a link to the dialog box that is opened by the command. |

# Command Listing

Use the links in this section to access Help topics that describe the commands that are available at the command line in AutoCAD Civil.

| Feature | Command Summary |
|---|---|
| Alignments | Alignment Command Reference (page 1016) |
| Assemblies and Subassemblies | Assembly and Subassembly Command Reference (page 1421) |
| Corridors | Corridor Command Reference (page 1375) |
| Drawings | Drawing Command Reference (page 124) |
| Drawing and Data Sharing | Drawing Sharing Command Reference (page 1663) |
| Google Earth Import/Export | Google Earth Import and Export Command Reference (page 1671) |
| Grading | Grading Command Reference (page 804) |
| Intersections | Intersection Command Reference (page 1460) |
| LandXML | LandXML Command Reference (page 1683) |
| Lines/Curves | Lines and Curves Command Reference (page 597) |
| Materials Analysis | Material Analysis Command Reference (page 1180) |
| Parcels | Parcels Command Reference (page 853) |
| Pipes | Pipe Network Command Reference (page 1284) |
| Plan Production | Plan Production Tools Command Reference (page 1725) |
| Points | Point Command Reference (page 516) |
| Profile Views | Profile View Command Reference (page 1121) |
| Profiles | Profile Command Reference (page 1121) |
| Project Management | Project Management Command Reference (page 182) |
| Sections | Sections Command Reference (page 1158) |
| Sites | Sites Command Reference (page 728) |
| Surfaces | Surfaces Command Reference (page 715) |
| Survey | Survey Command Reference (page 391) |
| Utilities | Utilities Command Reference (page 1651) |
| Transparent commands | Transparent Command Reference (page 1600) |

| Feature | Command Summary |
|---|---|
| User Interface | User Interface Command Reference (page 107) |

# AutoCAD Commands and AutoCAD Civil Objects

The following topic provides information about AutoCAD command operations on certain AutoCAD Civil objects and the results.

All AutoCAD Civil objects support the AutoCAD List, Explode, and Erase commands.

**NOTE** Exploding an object results in an unnamed AutoCAD block. Exploding again results in a base AutoCAD entity.

Most AutoCAD Civil objects support other commands to some degree. These commands include Copy, Move, Mirror, Offset, Scale, Block/wBlock, Rotate, Trim and Extend.

**NOTE** You can get the "Copy" behavior by using the Home tab ➤ Modify panel ➤ Copy ⬚ command on the ribbon, or by entering COPY at the command line or by using the Basic Modify Tools ➤ Copy command on the shortcut menu. You can get the "Block/Wblock" behavior by using the Clipboard ➤ Copy or Clipboard ➤ Copy With Base Point commands on the shortcut menu, pressing Ctrl + C, or entering WBLOCK at the command line.

## Supported AutoCAD Command Notes

Surface
If the surface is erased or exploded, profiles and sections that reference that surface become static.
Mirror will create a copy but will not honor the mirror settings.
Point groups used in the surface definition are included as part of the Wblock.

Point
When Wblocking and Inserting points, or copying points, a dialog box is displayed to resolve number conflicts.

**Table**
Wblocking a table also copies the required geometry. For example, the points are copied along with the point table.

Alignment
Exploding or erasing an alignment explodes or erases all objects that are dependent on the alignment (such as profiles, profile views, sample lines, sections, and section views).
Copying the alignment will create an alignment with a new name in the same site. All dependent objects will be copied and included with the new alignment.
Wblock/Insert will copy the alignment and all dependent objects into a new site.

Parcel (select parcel label)
When copying or Wblocking a parcel, the dependent parcel segments are included. If the objects are inserted into the same drawing, a new site is created.
Only the parcel labels can be moved.

Parcel Segment Lines
Wblocking/Inserting the parcel segment lines creates segment lines in a new site.

Grading

There are two distinct copy operations which can be done on a grading object. Copying the footprint feature line or individual target line creates a new feature line in a new site. Copying the entire grading object copies the footprint, target, and criteria.

The footprint can be moved. The target line cannot be moved.

If the grading criteria references a surface, the surface will be included in a Wblock/Insert operation.

Profile

Erasing a profile or section erases it from all profile or section views.

Profiles and sections can only be moved vertically within a view.

Profile View

If the profile or profile view is Wblocked, all objects required for the profile or profile view (such as the surface, alignment, profile, sample lines, and sections) are copied. If the objects are copied into the same drawing, a new site is created.

Sample Line

You cannot copy or move a Sample Line

Section

Erasing a profile or section erases it from all profile or section views.

Profiles and sections can only be moved vertically within a view.

If the sample line, section, or section view is Wblocked, all objects that are required for the sample line, section, or section view (such as the surface, alignment, sections, and sample lines) are copied. If the objects are copied into the same drawing, a new site is created.

Assembly

The names of assemblies and subassemblies are incremented when copied. Copying an assembly copies the assembly and all dependent subassemblies.

Mirror will create a copy but will not honor the mirror settings.

Corridor

Copying or Wblocking a corridor copies all of the dependent data such as assemblies, alignments, and surfaces.

Pipe Network

Network Objects are not accessible through Object Enabler environment.

Pipe

Copying a network part profile will create a copy of the parent network part (pipe or structure).

Structure

Copying, mirroring, arraying, and offsetting pipes and structures updates the references of any connected parts.

Copied pipes and structures will be in the same network.

Moving or rotating a structure causes any connected parts to move with it. (the connections are maintained).

Moving or rotating a pipe breaks the connection to any connected parts.

Scaling a pipe or structure affects position but does not affect size.

Wblocking pipes and structures copies all dependent objects.

**Survey Network**

Using the copy command on a survey network creates feature lines. The move command disassociates the figure from the model.

**Survey Figure**

Using the offset command on a survey figure generates a feature line object, not a survey figure.

The move commands dissassociates the figure from the model.

**Mass Haul Line**

Using the explode command keeps the layer and display settings.

With the Extend command, the object can be selected as an edge to extend to, but cannot be selected to extend

With the Trim command, the object can be selected as a trim edge, but cannot be trimmed

If the mass hauls is Wblocked, all objects that are required for it will be copied. Wblock/Insert is used when the user keys in ^C and ^V.

**Mass Haul View**

Views are automatically named according to "Existing Name" (#), where # is an increment. If a conflict is found, the next counter will be used.

The copy command or right-click "Copy Selection" will create a copy of the View with a name of "Existing Name" (#), where # is an increment.

The explode command will keep the layer and display settings. Exploding the View will not explode the haul line.

The mirror command is successful on a View, but it will not honor the mirror settings. You will get an additional copy of the Profile View.

If the View is Wblocked, all objects that are required for the View will be copied. Wblock/Insert is used when the user keys in ^C and ^V

**Civil Projection in Profile View and Section View**

Erase will remove from view but should not remove plan object.

Explode will keep the layer and display settings.

**AutoCAD Projection in Profile View and Section View**

Erase will remove from view but should not remove plan object.

Explode will keep the layer and display settings.

**Intersection**

Explode will keep the layer and display settings.

**Building Site**

Explode will keep the layer and display settings.

# Dialog Box Reference

# Alignment Dialog Boxes

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The following topics provide information about the Alignment dialog boxes.

## Alignment Entities Vista

Use this vista to view and edit entities that are part of the solved alignment geometry.

Use the Alignment Entities window to navigate through all solved alignment entities. You can select an entity in the Alignment Entities window and then edit it in the Alignment Layout Parameters window. Attributes in the Alignment Entities window are automatically updated as you edit the alignment.

You can filter the display to show a selected range of entities along an alignment. See Selecting Alignment Entities (page 989) for more information.

The Alignment Entities vista is like a spreadsheet. Each row represents an alignment entity and each column represents an attribute for that entity. You can resize, move, and copy columns.

The first column (No.) is always displayed. It does not move when you use the scroll bar.

**NOTE** Two column configurations are available: default and criteria-based design. To switch between the configurations, or to create your own configuration, right-click a column heading, and click Customize Columns.

This window remains open when you grip edit the alignment or enter another command. The window closes when you either delete the currently selected alignment, click the X button, or click ▣ on the Alignment Layout Toolbar (page 1747).

No.
Specifies the entity number in the order of selection. This variable changes as entities are selected and deselected.

A ⚠ warning symbol is displayed in this cell if the sub-entity violates the specified design criteria (page 866).

Number
Specifies the entity number in the order in which it appears along the alignment. This value changes as alignment entities ahead of it are created or deleted. This column cannot be hidden or deleted.

Curve Group Index
Specifies the curve group description.

**Curve Group Sub-Entity Index**

Specifies the curve group sub-entity description.

**Type**

Specifies either line, curve, or spiral as the entity type.

**Length**

Specifies the length of the entity.

A ⚠ warning symbol is displayed in this cell if the sub-entity length value is not greater than or equal to the value in the Minimum Length cell.

**Direction**

Specifies the direction of the entity.

**Minimum Length**

Specifies the minimum transition length defined in the design criteria file (page 2472) for the current design speed.

**Radius**

Specifies the radius of the curve.

A ⚠ warning symbol is displayed in this cell if the sub-entity radius value is not greater than or equal to the value in the Minimum Radius cell.

**Minimum Radius**

Specifies the minimum radius value defined in the design criteria file (page 2472) for the current design speed.

**A**

Specifies the A value.

**Constraint1**

Specifies the constraint of the entity: either fixed, free, or floating.

**Constraint2**

Specifies the number of points in the constraint of the tangent.
If the entity was created by best fit, [...] appears to the right of the Best Fit entry. If the entity has been

edited since it was created from best fit regression data, ▽ appears. Click [...] to open the Regression Data vista (page 1990) and review the regression points and synchronize the entity to the regression data.

**Bearing**

Specifies the direction of the entity.

**Start Station**

Specifies the starting station for the entity.

**End Station**

Specifies the ending station for the entity.

**Start Point**

Specifies the coordinates for the starting point of the entity.

**End Point**

Specifies the coordinates for the end point of the entity.

**Pass Through Point1**

Specifies the point coordinates for the first pass-through point of an alignment entity.

**Pass Through Point2**

Specifies the point coordinates for the second pass-through point of an alignment entity.

**Pass Through Point3**

Specifies the direction of the third user-specified pass-through point on a curve entity.

**Spiral Type**

Specifies the spiral type.

**Incurve**

Specifies whether the curve is incoming or outgoing.

**Compound**

Displays True when a spiral has a curve at each end.

**Radius In**

Specifies the incoming curve radius.

**Radius Out**

Specifies the outgoing curve radius.

**Total X**

Specifies the tangent distance between the beginning point of the incoming spiral (TS) and the endpoint of the spiral and beginning point of the circular curve.

**Total Y**

Specifies the perpendicular offset distance between the beginning point of the circular curve (SC) and the incoming tangent.

**Short Tangent**

Specifies the distance from the spiral point of the horizontal tangent intersection (SPI) to the end of the spiral (SPI).

**Long Tangent**

Specifies the distance from the beginning of the spiral (TS) to the spiral point of horizontal tangent intersection (SPI).

**P**

Specifies the offset distance from the incoming tangent to the beginning point (PC) of the shifted circular curve.

**K**

Specifies the abscissa between the beginning point of the spiral (TS) and the beginning point of the shifted circular curve.

**Spiral Definition**

Specifies the spiral definition.

**SPI Station**

Specifies the station value of the SPI. Adds the tangential distance from the beginning of the Spiral (TS) to the SPI to the station value of the TS.

**SPI Northing**

Specifies the XY coordinates of the spiral point of horizontal tangent intersection.

**SPI Easting**

Specifies the XY coordinates of the spiral point of horizontal tangent intersection.

**SPI Included Angle**

Specifies the angle representing the difference in direction of the incoming tangent and the outgoing tangent of the spiral.

**Radial Point Northing**

Specifies the coordinates of the spiral northing.

**Radial Point Easting**

Specifies the coordinates of the spiral easting.

**Center Point**

Specifies the coordinate of the center point of a circular curve.

**Direction At Through Point1**

Specifies the XY coordinates of the first user-specified pass-through point.

**Direction At Through Point2**

Specifies the XY coordinates of the second user-specified pass-through point.

**Greater than 180 Degrees**

- Displays True if the curve angle is defined as greater than 180 degrees.

- Displays False if the curve angle is defined as less than 180 degrees.

**Compound**

Displays True when the spiral has a curve at each end.

**Delta Angle**

Specifies the angular difference in the direction of the incoming and out going tangents.

**External Tangent**

Specifies the distance from the beginning of the curve (PC) to the PI or from the PI to the end of the curve (PT).

**Chord Direction**

Specifies the bearing along the line joining the beginning of the curve (PC) and the end of the curve (PT).

**External Secant**

Specifies the radial difference from the PI to the midpoint of the curve.

**PI Included Angle**

Specifies the included angle between the incoming and outgoing tangents of the curve.

**Chord Length**

Displays the chord length of the circular curve.
Specifies the distance along the line joining the beginning of the curve (PC) and the end of the curve (PT).

**Mid-Ordinate**

Specifies the distance from the midpoint of the curve, perpendicular to the midpoint of the chord.

**PI Station**

Specifies the station value of the PI by adding the PC to the PI tangent length to the station value at the beginning of the curve (PC).

**Design Speed**

Specifies the design speed for each sub-entity.

Minimum Radius Table

Specifies the minimum radius table for each sub-entity. The Minimum Radius Table is defined in the design criteria file (page 2472).

A ⚠ warning symbol is displayed in this cell if the sub-entity radius value is not greater than or equal to the value in the Minimum Radius cell.

Transition Length Table

Specifies the minimum transition length table for each sub-entity. The Transition Length Table is defined in the design criteria file (page 2472).

A ⚠ warning symbol is displayed in this cell if the sub-entity length value is not greater than or equal to the value in the Minimum Length cell.

Attainment Method

Specifies the superelevation attainment method for each sub-entity. The Attainment Method is defined in the design criteria file (page 2472).

**Related procedures:**

■ Criteria-Based Alignment Design (page 866)

■ Selecting Alignment Entities (page 989)

■ Editing Alignment Entity Layout Parameters (page 994)

# Alignment Geometry Points Dialog Box

Use this dialog box to specify the alignment geometry points that you want to label.

## Select Geometry Points To Label

☑ Select All

Selects all check boxes in the Label column.

☐ Deselect All

Clears all check boxes in the Label column.

Geometry Points Column

Identifies the available geometric details.

**NOTE** The Abbreviations (page 1855) tab in the Drawing Settings dialog box also lists these geometry points.

Label Column

Specifies whether to label a geometry point.

**Related procedures:**

■ Using Label Sets (page 973)

# Alignment Labels Dialog Box

Use this dialog box either to create and save label sets or to import an existing label set.

**Type**

Specifies the type of label, which is related to elements in the alignment that can be labeled. After you select a label type, you assign a style. Select one or more label types to include in the label set:

- Major station

- Minor station

- Geometry points

- Station equations

- Design speeds

- Profile Geometry Points

- Superelevation Critical Points

**NOTE** You can add a label type multiple times in a label set. For example you could add geometry points twice and then apply a style to the first entry of geometry points at specified starting and ending stations. Then, in the second entry for geometry points, you could specify another style and the starting and ending stations where you want to apply that style.

**<Label Type> Style List**

Lists the styles available for the specified label type. Select a style from the list.

**Style Selection**

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

**Style Detail**

Opens the Style Detail dialog box. Preview the style and creation information.

**Add>>**

Adds the selected element and style to the label set.

**NOTE** You must first select major stations in order to add minor stations.

Deletes the selected element in the label set.

**Reset**

Resets manual or grip edits that may have been made to any labels.

**Type**

Displays the label types added from the Type list.

**Style**

Displays the label style selected and added from the Style list. Click to change the style.

**Increment**

Displays the increment at which the label is displayed. To change the default value, click in the cell, and enter a new value.

**NOTE** Increment does not apply to all label types.

**Start Station**

Displays the starting station at which the label is displayed. By default, the start station is at the beginning of the alignment. To change the default display, clear the check box and enter a new value.

**End Station**

Displays the ending station at which the label is displayed. By default, the end station is at the end of the alignment. To change the default display, clear the check box and enter a new value.

**Geometry Points To Label**

Specifies the geometry points to label for the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types. You can label a selection of geometry points in a label type using a selected style. You can then add another instance of the same label type, which can label a second selection of

geometry points using a different style. Click [...] to select the geometry points to label.

**Profile**

For Profile Geometry Points label types, specifies which profile's geometry points are labeled on the

alignment [...] to change the profile.

**Station Index Increment**

When a station label style is in Station Index Format, this is the base value used to format the station labels.

For example, when station labels are displayed at 20-meter increments using the default Station Format, they are formatted as 0+00, 0+20, 0+40, 0+60, etc. If the same labels used the Station Index Format, with the Station index Increment set to 10, the labels would be formatted as 0+0, 2+0, 4+0, 6+0, etc. Changing the Station Index Increment to 20 would format the labels as 0+0, 1+0, 2+0, 3+0, etc.

**Import Label Set**

Opens the Select Style Set dialog box. Select a previously saved label set style and use it on the current alignment. Importing a label set overwrites the existing setting on the Labels tab.

**Save Label Set**

Opens the Label Set Dialog Box (page 1771). Configure the name and other settings for the current label set. This set is saved on the Toolspace Settings tab.

**Related procedures:**

■  Using Alignment Label Sets (page 973)

# Alignment Layout Parameters Window

Use this window to display and edit many attributes of any given single sub-entity within an alignment.

You can use the Alignment Layout Parameters window to change individual lines, curves, and spirals in the alignment. Use this window along with the Alignment Entities vista, which displays all of the sub-entities that make up a given alignment.

You can edit available parameters. You cannot edit parameters that are unavailable (shaded).

If the alignment uses design criteria, then the Alignment Layout Parameters window contains up to three

panels that can be collapsed by clicking ⌃ . If design criteria has not been associated with the alignment, then only the Layout Parameters panel is available.

You cannot dock the Alignment Layout Parameters window, but you can pin or resize it.

This window stays open when you grip edit the alignment or enter another command. The window closes when you either delete the currently selected alignment, click the X button in the upper right-hand corner of the window, or click  on the Alignment Layout Tools toolbar (page 1747).

**Design Speed**

Specifies the design speed of the current sub-entity.

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**NOTE** If multiple speeds are assigned to a sub-entity, then the highest speed is used to look up constraint values from the design criteria file and to validate the design checks. This ensures the safest design for that sub-entity.

---

### Design Criteria Panel

The following parameters are displayed only if design criteria has been associated with the alignment.

Use the Design Criteria panel to apply design criteria that is different from the default values that have been set for the alignment. To change a criterion value, click the Value cell in the appropriate row.

A  warning symbol in the Property column indicates that the sub-entity design violates the criteria set in the design criteria file. You can find and correct the specific parameter that has been violated by examining the Layout Parameters panel.

Minimum Radius Table

Defines the minimum radii for a given road type and design speed.

Transition Length Table

A table of values in the Superelevation Attainment Method formulas. You can use the table to calculate the distances between the critical superelevation transition points for different types of roadways as a function of curve radius and design speed.

Attainment Method

Specifies how superelevation is applied, and the method (crowned or planar) that is used to calculate superelevation transition stations for different types of roadways.

### Layout Parameters

Use this panel to examine and change the general layout parameters of the selected sub-entity.

Click  Show More to expand the number of parameters that appears in the window. The default, collapsed view displays editable and other important parameters. Use the horizontal and vertical scroll bars when the list is longer or wider than the window.

When a design criteria file has been associated with an alignment, the Constraints column displays the value to which a given parameter is limited. The selected design criteria file defines the constrained values. When a parameter value violates the range displayed in the Constraints column, a  warning symbol appears next to the parameter name.

---

**NOTE** The constraint definition for each alignment entity determines the parameters that are displayed in the Layout Parameters panel.

---

### Line

Entity

Specifies a number for the Line, in the order of creation.

Curve Group Index

Specifies the curve group description.

Curve Group Sub-Entity Index
Specifies the curve group sub-entity description.

Type
Specifies either Line, Curve, or Spiral as the sub-entity type.

Constraint1
Specifies either Fixed, Free, or Floating as the general constraint type.

Constraint2
Specifies a detailed description of the entity constraints.

Length
Specifies the Line length.

> **NOTE** You can edit the length of a floating line that is attached to the end of an entity.

Bearing
Specifies the direction of the line.

Start Station
Specifies the starting station of the line for the solved portions of the alignment.

End Station
Specifies the end station of the Line for the solved portions of the alignment.

Start Point
Specifies the XY coordinates for the start point of the Line.

End Point
Specifies the XY coordinates for the endpoint of the Line.

Pass Through Point1
Specifies the XY coordinates of the first user-specified pass-through point.

Pass Through Point2
Specifies the XY coordinates of the second user-specified pass-through point.

## Curve (independent or in SCS)

Entity
Specifies the curve number relative to the order of creation.

Curve Group Index
Specifies the curve group description.

Curve Group Sub-Entity Index
Specifies the curve group sub-entity description.

Type
Specifies Curve as the entity type.

Constraint1
Specifies either Fixed, Free, or Floating as the general constraint type.

Constraint2
Specifies a detailed description of the entity constraints.

Radius
Specifies the radius of the curve entity.

**Length**

Specifies the length of the curve entity.

**Pass Through Point1**

Specifies the XY coordinates of the first user-specified pass-through point.

**Pass Through Point2**

Specifies the XY coordinates of the second user-specified pass-through point.

**Pass Through Point3**

Specifies the XY coordinates of the third user-specified pass-through point.

**Center Point**

Specifies the XY coordinates of the center point.

**Start Station**

Specifies the starting station of the curve for the solved portion of the alignment. Unsolved curves do not display a Start Station.

**Start Point**

Specifies the XY coordinates of the start point of the curve.

**End Station**

Specifies the end station of the curve for solved portions of the alignment. Unsolved curves do not display an end station.

**End Point**

Specifies the XY coordinates of the endpoint of the curve.

**Direction At Through Point 1**

Specifies the XY coordinates of the first user-specified pass-through point.

**Direction At Through Point 2**

Specifies the XY coordinates of the second user-specified pass-through point.

**Greater Than 180 Degrees**

- Displays True if the curve angle is defined as greater than 180 degrees.

- Displays False if the curve angle is defined as less than 180 degrees.

**Compound**

- Displays True if the curve is defined as compound to the curve before it.

- Displays False if the curve is defined in the reverse direction to the curve before it.

**Delta Angle**

Specifies the included angle of the solved portion of the curve.

**External Tangent**

Specifies the distance from either the beginning of the curve (PC) to the point of intersection (PI), or from the PI to the end of the curve (PT).

**Chord Direction**

Specifies the bearing along the line joining the beginning of the curve (PC) and the end of the curve (PT).

**External Secant**

Specifies the radial difference from the PI to the midpoint of the curve.

PI Included Angle

Specifies the included angle between the incoming and outgoing tangents of the curve.

Chord Length

Specifies the distance along the line joining the beginning of the curve (PC) and the end of the curve (PT).

Mid-Ordinate

Specifies the distance from the midpoint of the curve, perpendicular to the midpoint of the chord.

PI Station

Specifies the station value of the PI by adding the PC to the PI tangent length to the station value at the beginning of the curve (PC).

## Spiral

Entity

Specifies the spiral number relative to the order of creation.

Curve Group Index

Specifies the curve group description.

Curve Group Sub-Entity Index

Specifies the curve group sub-entity description.

Type

Specifies Spiral as the entity type.

Constraint1

Specifies the spiral as either Fixed or Free.

Constraint2

- Specifies the Spiral In, Radius, and Spiral Out for free spirals that are part of a Spiral-Curve-Spiral group.

- Specifies the start point, direction, radius, and length for fixed spirals.

Spiral Type

Displays True when the spiral has a curve at both ends.

Start Station

Specifies the Start Station at the beginning of the spiral.

Start Point

Specifies the XY coordinates of the Start Point of the spiral.

End Station

Specifies the end station for the solved spiral portions of the alignment. Unsolved portions do not display an end station.

End Point

Specifies the XY coordinates of the End Point of the spiral.

Incurve

Specifies the incoming curve radius.

Length

Specifies the length of the Spiral.

A

Specifies the A value.

Compound

- Displays True if the curve is defined as compound to the curve before it.

- Displays False if the curve is defined in the reverse direction to the curve before it.

Radius In

Specifies the incoming curve radius.

Radius Out

Specifies the outgoing curve radius.

Total X

Specifies the tangent distance between the beginning point of the incoming spiral (TS) and the endpoint of the spiral and beginning point of the circular curve.

Total Y

Specifies the perpendicular offset distance between the beginning point of the circular curve (SC) and the incoming tangent.

Short Tangent

Specifies the distance from the spiral point of the horizontal tangent intersection (SPI) to the end of the spiral (SPI).

Long Tangent

Specifies the distance from the beginning of the spiral (TS) to the spiral point of horizontal tangent intersection (SPI).

P

Specifies the offset distance from the incoming tangent to the beginning point (PC) of the shifted circular curve.

K

Specifies the abscissa between the beginning point of the spiral (TS) and the beginning point of the shifted circular curve.

Spiral Definition

Specifies the spiral definition.

PI Included Angle

Specifies the included angle between the incoming and outgoing tangents of the curve.

SPI Station

Specifies the station value of the SPI. Adds the tangential distance from the beginning of the Spiral (TS) to the SPI to the station value of the TS.

SPI Northing

Specifies the XY coordinates of the spiral point of horizontal tangent intersection.

SPI Easting

Specifies the XY coordinates of the spiral point of horizontal tangent intersection.

SPI Included Angle

Specifies the angle representing the difference in direction of the incoming tangent and the outgoing tangent of the spiral.

Radial Point Northing

Specifies the coordinates of the spiral northing.

Radial Point Easting

Specifies the coordinates of the spiral easting.

**Design Checks**

The Design Checks panel displays the name and contents of the design check set that is associated with the alignment.

When the sub-entity design violates a design check, a ⚠ warning symbol appears next to the affected design check.

**NOTE** The Design Checks panel is available only for alignments that use a design check set.

**Related procedures:**

■ Editing Alignment Entity Layout Parameters (page 994)

■ Criteria-Based Alignment Design (page 866)

# Alignment Layout Tools

Use this toolbar to draw simple tangent-tangent lines, create constraint-based alignment geometry, and edit an alignment.

The name of the currently selected alignment is displayed at the top of the toolbar. When you click an icon, the current command is displayed at the bottom of the toolbar. The toolbar remains open when you grip edit the alignment or enter another command. The toolbar closes when you either delete the currently selected alignment or click the X button in the upper-right corner of the toolbar.



Tangent-Tangent (No Curves)

Adds a series of fixed tangents between specified points.

Tangent-Tangent (With Curves)

Adds a series of fixed tangents between specified points, with free curves automatically added at the points of intersection.

Curve And Spiral Settings

Specifies the curve parameters to use with the Tangent-Tangent With Curves command.

Insert PI

Breaks a fixed line into two adjacent fixed lines by creating a point of intersection (PI) at a point that you specify.

Delete PI

Creates a single tangent from two adjacent tangents by removing their point of intersection (PI).

Break Apart PI

Separates the point of intersection (PI) where the endpoints of two fixed or floating tangents meet.

**Convert AutoCAD Line And Arc**

Creates a fixed two-point line or three-point curve alignment entity from an AutoCAD object.
An alignment sub-entity that has been converted from an AutoCAD entity may be added as a solved portion of the alignment geometry in either of two ways:

■ Before the AutoCAD entity is converted, it must be attached to an unattached end point of another solved sub-entity in the alignment.

■ After the AutoCAD entity has been converted, it may be joined to the solved alignment geometry using the alignment layout tools.

**Reverse Sub-entity Direction**

Reverses the direction of a fixed, unconnected line or curve sub-entity.

**Delete Sub-Entity**

Deletes a specified alignment sub-entity.

**Edit Best Fit Data For All Entities**

Displays a table of data that contains the original regression data for all alignment sub-entities that were created by best fit.

**Pick Sub-entity**

Displays a selected sub-entity's parameters for editing.

**Alignment Layout Parameters**

Displays a vertical table of numeric data about a single, selected alignment sub-entity.

**Alignment Entities**

Displays a horizontal table of numeric data about multiple, selected alignment subentities.

**Undo**

Reverses the last AutoCAD Civil or AutoCAD command.

**Redo**

Reverses the last Undo command. Redo is limited to one operation.

## Line Tools

Add constraint-based fixed, free, or floating lines to an alignment.

**Fixed Line (Two points)**

Adds a fixed line between two specified points.

**Fixed Line (From curve end, length)**

Adds a fixed line to and from the endpoint of an existing curve to another specified point. Tangency is not maintained if either entity is edited.

**Fixed Line - Best Fit**

Adds the most probable fixed line through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

**Floating Line (From curve, through point)**

Adds a floating line from any point on an existing curve entity through a specified point.

**Floating Line (From curve end, length)**

Adds a floating line tangent, with a specified length, to the end of a curve entity. Tangency is maintained to the attached entity end, regardless of how the entity is edited.

**Floating Line - Best Fit**

Adds the most probable floating line from a point on an existing entity through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entity, regardless of how the entity is edited.

**Free Line (Between two curves)**

Adds a free line between two existing curves.

## Curve Tools

Add constraint-based fixed, free, or floating curves to an alignment.

**Fixed Curve (Three points)**

Adds a fixed curve through three points.

**Fixed Curve (Two points and direction at first point)**

Adds a fixed curve that is defined by specified start and end points and a direction at the start point.

**Fixed Curve (Two points and direction at second point)**

Adds a fixed curve that is defined by specified start and end points and a direction at the end point.

**Fixed Curve (Two points and radius)**

Adds a fixed curve that is defined by specified radius, direction, and start and end points.

**Fixed Curve (From entity end, through point)**

Adds a fixed curve from the end of an existing entity to a specified end point.

**Fixed Curve (Center point and radius)**

Adds a full, fixed circle that is defined by a specified center point, direction, and radius.

**Fixed Curve (Center point and through point)**

Adds a full, fixed circle that is defined by a specified center point, direction, and pass-through point.

**Fixed Curve (Through point, direction at point and radius)**

Adds a full, fixed circle that is defined by a specified pass-through point, direction at the pass-through point, curve direction, and radius.

**Fixed Curve - Best Fit**

Adds the most probable fixed curve through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

**Floating Curve (From entity, radius, through point)**

Adds a floating curve, which is defined by a specified radius and angle range, from an existing entity to a specified end point.

**Floating Curve (From entity end, through point)**

Adds a floating curve from the end of an existing entity to a specified pass-through point.

**Floating Curve (From entity, through point, direction at point)**

Adds a floating curve from an existing entity to a specified pass-through point.

**Floating Curve (From entity end, radius, length)**

Adds a floating curve, which is defined by a specified direction, radius, and length, to the end of an existing entity.

**Floating Curve - Best Fit**

Adds the most probable floating curve from an existing entity through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entity, regardless of how the entity is edited.

**Free Curve Fillet (Between two entities, radius)**

Adds a free curve, which is defined by a specified angle range and radius, between two entities.

**Free Curve Fillet (Between two entities, through point)**

Adds a free curve, with a specified pass-through point, between two entities.

**Free Curve - Best Fit**

Adds the most probable free curve between two existing entities, and through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entities, regardless of how the entities are edited.

### Lines with Spiral Tools

**Floating Line with Spiral (From curve, through point)**

Adds a floating spiral-line group, which is defined by a specified pass-through point, to a curve.

**Floating Line with Spiral (From curve end, length)**

Adds a floating spiral-line group, which is defined by the line length, to a curve.

### Curves with Spiral Tools

**Floating Curve with Spiral (From entity end, radius, length)**

Adds a floating spiral-curve group, which is defined by a specified radius and pass-through point, to a line.

**Floating Curve with Spiral (From entity, radius, through point)**

Adds a floating spiral-curve group, which is defined by a specified radius and length, to a curve.

**Floating Reverse Curve with Spirals (From curve, radius, through point)**

Adds a floating reverse spiral-spiral-curve group, which is defined by a specified radius and pass-through point, to a curve.

**Floating Reverse Curve with Spirals (From curve, two points)**

Adds a floating reverse spiral-spiral-curve group, which is defined by two specified pass-through points, to an existing curve.

**Free Spiral-Curve-Spiral (Between two entities)**

Adds a free spiral-curve-spiral group between

- Two tangents, creating a simple spiral.

- A tangent and a curve, creating a compound spiral at one end and a simple spiral at the other end.

- Two curves, creating two compound spirals at each end.

**Free Compound Spiral-Curve-Spiral-Curve-Spiral (Between two tangents)**

Adds a free compound spiral-curve-spiral-curve-spiral group between two tangents. You can specify a zero length for any of the three spirals.

**Free Reverse Spiral-Curve-Spiral-Spiral-Curve-Spiral (Between two tangents)**

Adds a free reverse spiral-curve-spiral-spiral-curve-spiral group between two tangents.

## Spiral Tools

**Fixed Spiral**

Adds a fixed spiral, which is defined by a specified radius and length, to the end of a line or curve.

**Free Spiral (Between two entities)**

Adds a free, compound spiral between two curves with different radii.

**Free Compound Spiral-Spiral (Between two curves)**

Adds a free compound spiral-spiral group between two curves.

**Free Reverse Spiral-Spiral (Between two curves)**

Adds a free reverse spiral-spiral group between two curves.

**Free Compound Spiral-Spiral (Between two tangents)**

Adds a free compound spiral-spiral between two tangents.

**Free Compound Spiral-Line-Spiral (Between two curves, spiral lengths)**

Adds a free compound spiral-line-spiral group, which is defined by specified spiral lengths, between two curves.

**Free Reverse Spiral-Line-Spiral (Between two curves, spiral lengths)**

Adds a free reverse spiral-line-spiral group, which is defined by specified spiral lengths, between two curves.

**Free Compound Spiral-Line-Spiral (Between two curves, line length)**

Adds a free compound spiral-line-spiral group, which is defined by a specified line length, between two curves.

**Free Reverse Spiral-Line-Spiral (Between two curves, line length)**

Adds a free reverse spiral-line-spiral, which is defined by a specified line length, between two curves.

**Related procedures:**

- Adding Lines, Curves, and Spirals to an Alignment (page 898)

# Alignment Properties Dialog Box

Use this dialog box to review or edit many alignment details such as the name, design speeds, label sets, and superelevation.

Some of the tabs on this dialog box are conditional, subject to alignment type. For example, the Curb Return Parameters tab appears only for curb return alignments.

**See also:**

■ Alignment Properties (page 863)

## Information Tab (Alignment Properties Dialog Box)

Use this tab to change the alignment name, description and alignment style information, and to review details, such as when the style was most recently modified.

Name

Specifies the name of the current alignment style.

Description

Specifies an optional description of the current alignment style.

Object Style

Specifies the alignment style. Select other styles from the list.

 Style Selection

Specifies the style options. You can create a style, copy or edit the current style selection, or pick a style from drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Type

Specifies the alignment type: Centerline, Offset, Curb Return, or Miscellaneous.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Station Control Tab (Alignment Properties Dialog Box)

Use this tab to establish the Station Reference Point and assign station equations along the alignment.

**Reference Point Station**

X

Specifies the X coordinate value for the Station Reference Point. By default, this value corresponds to the starting point on the alignment. Enter a value or pick a point in the drawing.

Y

Specifies the Y coordinate value for the Station Reference Point. By default, this value corresponds to the starting point on the alignment. Enter a value or pick a point in the drawing.

 Pick Reference Point

Specifies the XY coordinates for the Station Reference Point. Pick points in the drawing to specify the coordinates.

Station

Specifies the station at the reference point.

**Station Information**

Start

Displays the beginning station for the first solved entity in the alignment.

End

Displays the ending station for the last solved entity in the alignment.

Length

Displays the length of the alignment as the summation of the length of solved entities along each alignment.

**Station Equations**

Add Station Equation

Specifies the location for a station equation. Click the button and specify the station for the new equation by either picking a station in the drawing or entering a raw station value on the command line.

Delete Station Equation

Deletes the selected station from the drawing. Select the station row that you want to delete and click this button.

Equation

Displays the index number of the equation sorted by raw station location on the alignment.

Raw Station Back

Displays the station value at the location of the Station Equation before any Station Equation values are assigned. If there is only one Station Equation value, then the Raw Station Back and Station Back values match.

Station Back

Displays the station value at the location of the Station Equation value before any Station Equations are assigned. If there is only one Station Equation value, then the Raw Station Back and Station Back values match.

If the alignment contains more than one Station Equation value, then the Station Back value of all Station Equation values, except for the first equation, are relative to the Station Ahead value and the Station Equation value directly before it.

Station Ahead

Specifies the new station values immediately after the Station Equation value.

Increase/Decrease

Specifies whether the stations increase or decrease in value following the Station Equation location.

Comment

Specifies notes or instructions relevant to the station equations. Enter optional comments. These comments can be used in labels for station equations.

## Masking Tab (Alignment Properties Dialog Box)

Use this tab to suppress the display of an alignment and its labels for a range of stations.

Add Masking Region

Click to add a new mask region. You are asked to specify the start and end stations for the region.

 Delete Masking Region

Click to remove a mask region selected in the table.

**Masking Table**

Region

Identifies the mask region along the alignment.

Mask

When selected, the mask is applied; the mask region does not display. When deselected, the mask is removed; the region displays. A deselected mask remains in the table for future use.

Lock to Start

Click to set the start of the first mask region at the start of the alignment.

Start Station

Displays the alignment station at which the mask region begins.

Lock to End

Click to set the end of the last mask region at the end of the alignment.

End Station

Displays the alignment station at which the mask region ends.

Comment

Use this space to add text notes about the mask region.

# Design Criteria Tab (Alignment Properties Dialog Box)

Use this tab to establish the design criteria, including speeds along the length of the alignment.

**Design Speeds**

Alignments can have an unlimited number of design speeds, but they can have only one design speed at any given location.

 Add Design Speed

Specifies the location for a new design speed. Click the icon and pick a point on the alignment or enter a value at the command line to create a design speed.

 Delete Design Speed

Deletes the selected design speed. Select the design speed row and click the button to delete the design speed.

Station

Displays the station location of the design speed.

Design Speed

Specifies the design speed for the location. Click in the cell to enter a new design speed or accept the default.

Comment

Specifies notes or instructions relevant to the design speeds. Enter optional comments. These comments can be used in labels.

**Criteria-Based Design (Unlabled)**

Use the following controls to change the design criteria file, default criteria, or design check set.

### Use Criteria-Based Design

Causes the Use Design Criteria File and Use Design Check Set check boxes to become available. If this check box is cleared, Design Speed is the only design criteria that will be applied to the alignment.

### Use Design Criteria File

Applies the default design criteria file to the alignment and causes the Default Criteria property table to become available. The first design criteria file found in the *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>* directory is used by default. Click

⊡ to select a different design criteria file.

The Default Criteria table displays the Properties, or table contents, that are defined in the selected design criteria file. Click the Value column to change a criteria table.

For more information, see Design Criteria Files (page 867).

### Use Design Check Set

Applies the default design check set to the alignment. Use the list to select from the available design check sets. You can also use the selector to create a design check set, copy or edit the current design check set, or pick a design check set from another alignment in the drawing.

For more information, see Alignment Design Checks (page 871).

## Profiles Tab (Alignment Properties Dialog Box)

Use this tab to view data about a profile associated with the alignment.

### Name

Specifies the name of the profile.

### Description

Specifies an optional description for the profile.

### Type

Specifies whether the current profile represents the existing ground or a finished grade.

### Surface Name

Displays the name of the surface from which the EG Profile gets elevations.

### Offset

Specifies the offset distance from the centerline of the parent horizontal alignment: either 0 for a centerline profile, a positive number for a right offset, or a negative number for a left offset.

### Update Mode

For existing ground profiles only. Specifies whether the profile updates automatically to reflect changes in surface elevation.

■ Dynamic: The profile automatically updates to reflect changes in the surface elevation or the geometry of the parent horizontal alignment.

■ Static: The profile shows surface elevations at the time of its creation. It does not update to reflect changes that occur later.

### Layer

Specifies the drawing layer on which the profile is placed.

### Style

Specifies the style used by the profile.

Station Start/End

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

Elevation Minimum/Maximum

Specifies the highest and lowest elevation values that occur along the profile.

## Profile Views Tab (Alignment Properties Dialog Box)

Use this tab to view data about a profile view.

Name

Specifies the name of the profile.

Description

Specifies an optional description for the profile.

Stations Start/End

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

Style

Specifies the style used by the profile.

Band Style

Specifies the band style used by the profile.

## Superelevation Tab (Alignment Properties Dialog Box)

Use this tab to enter and edit superelevation values and to add or delete transition stations.

### Superelevation Controls

Add A Transition Station

Specifies a transition station in the drawing.

Delete Transition Station

Deletes the selected transition station.

Set Superelevation Properties

Opens the Superelevation Specifications (page 1776) dialog box. Use this dialog box to define the settings and properties used to calculate the superelevation specifications for the alignment.

Hide Inside Lanes And Shoulders

If selected, hides the inside lanes and shoulders.

No.

Displays the entry number.

Superelevation Region

Displays the region number.

Station

Specifies the station of the superelevation transition point. Minimally, include the beginning and ending station of the alignment to enable establishing at least one superelevation zone.

Description

Specifies the type of transition point:

- Begin Alignment

- Begin Full Super

- Begin Normal Crown

- Begin Normal Shoulder

- End Alignment

- End Full Super

- End Normal Crown

- End Normal Shoulder

- Level Crown

- Low Shoulder Match

- Reverse Crown

- Shoulder Breakover

Left Side and Right Side Columns

Specifies the percent slopes for the various shoulder and lane segments.

Smoothing Curve Length

Specifies the length of the smoothing vertical curve at the transition point.

**Related procedures:**

- Adding Superelevation Specifications (page 884)

# Offset Parameters Tab (Alignment Properties Dialog Box)

For a dependent offset alignment, use this tab to set properties of the offset.

## Parent Alignment

Name

Specifies the name of the parent alignment, which controls the offset alignment.

Nominal Offset Value

Specifies the offset distance from the parent alignment.

Start Station – Lock to Start of Parent

Specifies the station of the parent alignment where the offset alignment begins. Enter a station value, or select the checkbox to begin the offset at the start of the parent alignment.

End Station – Lock to End of Parent

Specifies the station of the parent alignment where the offset alignment ends. Enter a station value, or select the checkbox to extend the offset to the end of the parent alignment.

Update Mode

Specifies whether the offset alignment reacts to edits of the parent alignment:

- **Dynamic.** The offset is updated with any changes to the parent alignment.

- ■ **Static**. The offset alignment geometry is preserved, unafffected by changes to the parent alignment.

Lock Mode

For dynamic update mode, specifies how the offset alignment is associated with the geometry of the parent alignment:
- ■ **Geometry Locking.** The start and end of the offset are located in relation to the nearest geometry points on the parent alignment.

- ■ **Station Locking**. The start and end of the offset are locked to the specified start and end stations on the parent alignment.

## Curb Return Parameters Tab (Alignment Properties Dialog Box)

For a curb return alignment, use this tab to review and edit design data.

### Parent Alignment 1

Name

Specifies the alignment that forms the basis of the design at the start of the curb return alignment.

Curb Return End Point Station

Specifies the station value on the parent alignment where the curb return alignment starts.

Curb Return Offset Value

Specifies the offset from the initial attachment point of the start of the curb return alignment, which is typically an offset alignment from the centerline. This field is editable.

### Parent Alignment 2

Name

Specifies the alignment that forms the basis of the design at the end of the curb return alignment.

Curb Return End Point Station

Specifies the station value on the parent alignment where the curb return alignment ends.

Curb Return Offset Value

Specifies the offset from the initial attachment point of the end of the curb return alignment, which is typically an offset alignment from the centerline. This field is editable.

Update Mode

Specifies whether the curb return alignment reacts to edits of the parent alignment or offset. This field is editable.
- ■ Dynamic. The curb return alignment is updated in response to edits of other intersection objects.

- ■ Static. The curb return alignment is at a fixed location, and does not respond to edits of other intersection objects.

Lock Mode

Specifies that the curb return is always geometry locked. The start and end of the curb return offset are located in relation to the nearest geometry points on the parent alignment.

# Alignment Style Dialog Box

Use this dialog box to control the display of each alignment component.

Create different styles to use in the various phases of your project. For example, you can create a first style to use specifically in the design phase that displays lines, curves, and other alignment subcomponents in

different colors. Then, you can create a second style to use for plotting that displays only the solved alignment geometry in one color and linetype.

**See also:**

- [Alignment Styles](#) (page 885)

# Information Tab (Alignment Style Dialog Box)

Use this tab to change the alignment style name and description information, and to review details about the alignment, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Design Tab (Alignment Style Dialog Box)

Use this tab to specify grip editing behavior. The value you enter increases or decreases the radius of curves to an even increment.

### Grip Edit Behavior

Enable Radius Snap

Specifies whether the cursor snaps to the specified increment when grip editing the radius of a free curve.

Radius Snap Value

Specifies the increment of the radius snap.

**NOTE** The snap value is based on zero. For example, if the current radius is 150.5 and the radius snap value is 10, when you grip edit the alignment to the next highest radius the radius value is 160, not 160.5.

# Markers Tab (Alignment Style Dialog Box)

Use this tab to specify the display and appearance of points on the alignment.

### Alignment Points and Marker Styles

Name

Displays the name of the point on the alignment.

Marker Style

Specifies the style for the point on the alignment. Click to open the Pick Marker Style dialog box where you can select a style or <None>.

### Arrowhead

Type

Specifies the arrowhead style for the alignment direction arrow. Select a style from the list or select User Arrow to select an AutoCAD block.

Size Options

Specifies the method used to determine the size of the arrowhead:

- Use Drawing Scale: Determines size of the arrowhead by multiplying the specified value by the drawing scale. Enter the scale factor.

- Use Fixed Scale: Activates the Fixed Scale options.

- Use Size in Absolute Units: Specifies that the arrowhead size is an absolute value based on the displayed units. Enter the value.

■ Use Size Relative to Screen: Specifies that the size of the arrowhead is a percentage of the drawing screen size. Enter the percentage.

Inches/Meters
Specify value for feet or meters.

Fit Options
Specifies how the arrow is displayed when the profile segment is shorter than the arrowhead. Select Shrink, Omit, or Always Draw.

Fixed Scale
Specifies independent fixed scale values when Size Options is set to Use Fixed Scale. Enter a value for X.

## Display Tab (Alignment Style Dialog Box)

Use this tab to change the display and visibility of alignment object components.

Alignment components include:

**Line**: Solved lines with true tangent lengths that are part of the alignment geometry.

**Curve**: Solved curves that are true curve lengths within the alignment.

**Spiral**: Spiral entities within the alignment geometry.

**Arrow**: Shows the direction of the alignment entities, either solved or unsolved.

**Line Extensions**: Line segments that extend beyond the solved portion of a line to a pass-through point or another constraint.

**Curve Extensions**: Curve segments that extend beyond the solved portion of a curve to a pass-through point or another constraint.

**Warning Symbol**: Markers that identify entities that violate the rules specified in the design criteria file. This component only applies to alignments that use design criteria.

**Pass-Through Points**: Markers that denote the pass-through points that define the geometry of any given entity.

**PI Points**: Markers that denote the pass-through points of two lines only when the pass-through points coincide exactly.

**Station Reference**: Markers that denote the station reference point on the alignment.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Alignment Style Dialog Box)

Use this tab to review all the information about the current alignment style. You can copy and paste this information to the clipboard.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

## Create Alignment From Objects Dialog Box

Use this dialog box to specify the attributes of an alignment, including the name, description, style, and starting station.

Name
Specifies the name of the alignment. Each alignment must have a unique name.

Type

Specifies the alignment type: Centerline, Offset, Curb Return, or Miscellaneous.

Description

Specifies an optional description of the alignment.

Starting Station

Specifies the station value that is assigned to the beginning of the first entity in the alignment.

NOTE Use the Station Control tab in the Alignment Properties dialog box to change the starting station after you create an alignment.

**General Tab**

Site

Specifies a site for the alignment. Select a site from the list, or click  to either create a site or select one from an object in the drawing. The alignment and the object you select in the drawing are associated with the same site.

NOTE The default selection is <None>, which places the alignment in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

Alignment Style List

Displays the current style. Click the arrow to select another alignment style in the drawing.

Style Selection

Specifies the style options. Create a style, copy or edit the current style selection, or pick a style from drawing.

Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Alignment Layer Display List

Displays the layer on which the alignment object will be created.

Object Layer

Opens the Object Layer Dialog Box (page 1985), in which you can select or create a layer on which the alignment will be drawn.

Alignment Label Set List

Displays the default alignment label set. You can either accept the default label set, or use the list to select a different one from the drawing.

Label Set Selection

Edits or copies the current label set, or creates a label set. The Style Detail icon previews the current style.

Style Detail

Opens the Style Detail dialog box. Preview the label set and creation information.

**Conversion Options**

Add Curves Between Tangents

Specifies whether to automatically add curves between each tangent.

- Selected: Curves are placed at each tangent-tangent intersection. The radius of the curve is calculated as a percentage of the tangent length and the deflection angle.

■ Cleared: Curves are not added automatically during creation.

---

**NOTE** You can add curves using the Alignment Layout Tools.

---

Erase Existing Entities

Specifies whether the entities you selected when creating the alignment are erased. Select the check box to erase the entities.

### Design Criteria Tab

Starting Design Speed

Specifies the design speed at the alignment starting station. If no other design speeds are applied to the alignment, then the Starting Design Speed is applied to the entire alignment.

Use Criteria-Based Design

Specifies whether design criteria is applied to the alignment. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

Use Design Criteria File

Specifies whether to associate a design criteria file with the alignment. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.
For more information, see Design Criteria Files (page 867).

Design Criteria File Selector

Specifies the design criteria file to associate with the alignment. The first design criteria file found in the following directory is used by default: *C:\Documents and Settings\All Users\Application*

*Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>*. Click ![button] to select a different design criteria file.

Default Criteria Table

Displays the standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

### Use Design Check Set

Use Design Check Set Check Box

Specifies whether to associate a design check set with the alignment. If this check box is cleared, then the Design Check Set list is not available.
For more information, see Alignment Design Check Sets (page 875).

Design Check Set List

Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

![icon] Design Check Set Selection

Edits or copies the current design check set, or creates a design check set.

### Related procedures:

■ Creating an Alignment from Graphic Entities (page 891)

■ Criteria-Based Alignment Design (page 866)

# Create Alignment - Layout Dialog Box

Use this dialog box to enter information about an alignment, such as its name, a description, and the starting station for the alignment. Also, you can use it to select an alignment style, label set style, site, and design criteria.

**Related procedures:**

- Creating an Alignment with the Alignment Layout Tools (page 887)
- Criteria-Based Alignment Design (page 866)

## General Tab (Create Alignment - Layout Dialog Box)

Use this tab to enter basic information about the alignment, including site, style, layer, and label set.

Name

Specifies the name of the alignment. Each alignment must have a unique name.

Description

Specifies an optional description of the alignment.

Starting Station

Specifies the station value that is assigned to the beginning of the first alignment entity created.

**NOTE** Use the Station Control tab in the Alignment Properties dialog box to change the starting station after you create an alignment.

### Site

Specifies a site for the alignment. Select a site from the Site list, or click  to either create a site or select one from an object in the drawing. The alignment and the object you select in the drawing are associated with the same site.

**NOTE** The default selection is <None>, which places the alignment in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

### Alignment Style

Alignment Style List

Displays the current style. Click the arrow to select another alignment style in the drawing.

 Style Selection

Specifies the style options. Create a style, copy or edit the current style selection, or pick a style from drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

### Alignment Layer

Alignment Layer Display List

Displays the layer on which the alignment object is created.

 Object Layer

Opens the Object Layer Dialog Box (page 1985), in which you can select or create a layer on which the alignment is drawn.

### Alignment Label Set

Alignment Label Set List
　Displays the default alignment label set. You can either accept the default label set, or use the list to select a different one from the drawing.

 Label Set Selection
　Edits or copies the current label set, or creates a label set. The Style Detail icon previews the current style.

 Style Detail
　Opens the Style Detail dialog box. Preview the label set and creation information.

# Design Criteria Tab (Create Alignment - Layout Dialog Box)

Use this tab to specify the design criteria settings, including the design criteria file and design check set.

Name
　Specifies the name of the alignment. Each alignment must have a unique name.

Description
　Specifies an optional description of the alignment.

Starting Station
　Specifies the station value that is assigned to the beginning of the first alignment entity created.

　**NOTE** Use the Station Control tab in the Alignment Properties dialog box to change the starting station after you create an alignment.

### Starting Design Speed

Specifies the design speed at the alignment starting station. If no other design speeds are applied to the alignment, then the Starting Design Speed is applied to the entire alignment.

### Use Criteria-Based Design

Specifies whether design criteria is applied to the alignment. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

### Use Design Criteria File

Use Design Criteria File
　Specifies whether to associate a design criteria file with the alignment. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.
　For more information, see Design Criteria Files (page 867).

Design Criteria File Selector
　Specifies the design criteria file to associate with the alignment. The first design criteria file found in the *C:\Documents and Settings\All Users\Application Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>* directory is used by default. Click  to select a different design criteria file.

Default Criteria Table
　Displays the alignment standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

### Use Design Check Set

Use Design Check Set Check Box
　Specifies whether to associate a design check set with the alignment. If this check box is cleared, then the Design Check Set list is not available.

For more information, see Alignment Design Check Sets (page 875).

Design Check Set List

Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

Design Check Set Selection

Edits or copies the current design check set, or creates a design check set.

# Create Offset Alignments Dialog Box

Use this dialog box to create dependent offset alignments from an existing alignment.

You can create multiple offsets in a single operation, including a different number on each side of the parent alignment. The offset distance can be different on each side of the parent alignment.

Alignment to Offset From

Specifies the name of the parent alignment.

Offsets Name Template

Specifies the format of the alignment name. If you want to change this format, click the button to open the Name Template Dialog Box (page 1806).

Station Range

Specifies the start and end stations on the parent alignment where the offset alignments will start and end. You can either enter numeric values, or click the buttons and pick locations in the drawing.

No. of Offsets on Left/Right

Specifies the number of offsets to create on each side of the alignment. Left and right sides are determined from a position facing toward the end of the parent alignment.

Incremental Offset on Left/Right

Specifies the offset distance on each side, between each pair of alignment.

**General Tab**

Site

Specifies a site for the alignment. Select a site from the list, or click   to either create a site or select one from an object in the drawing. The alignment and the object you select in the drawing are associated with the same site.

NOTE The default selection is <None>, which places the alignment in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

Alignment Style List

Displays the current style. Click the arrow to select another alignment style in the drawing.

Style Selection

Specifies the style options. Create a style, copy or edit the current style selection, or pick a style from drawing.

Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Alignment Layer Display List

Displays the layer on which the alignment object will be created.

 Object Layer

Opens the Object Layer Dialog Box (page 1985), in which you can select or create a layer on which to draw the alignment.

**Alignment Label Set List**

Displays the default alignment label set. You can either accept the default label set, or use the list to select a different one from the drawing.

 Label Set Selection

Edits or copies the current label set, or creates a label set. The Style Detail icon previews the current style.

 Style Detail

Opens the Style Detail dialog box. Preview the label set and creation information.

**Design Criteria Tab**

**Starting Design Speed**

Specifies the design speed at the alignment starting station. If no other design speeds are applied to the alignment, then the Starting Design Speed is applied to the entire alignment.

**Use Criteria-Based Design**

Specifies whether design criteria is applied to the alignment. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

**Use Design Criteria File**

Specifies whether to associate a design criteria file with the alignment. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.
For more information, see Design Criteria Files (page 867).

**Design Criteria File Selector**

Specifies the design criteria file to associate with the alignment. The first design criteria file found in the following directory is used by default: *C:\Documents and Settings\All Users\Application*

*Data\Autodesk\Civil2010\enu\Data\Corridor Design Standards\<units>*. Click  to select a different design criteria file.

**Default Criteria Table**

Displays the standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

**Use Design Check Set**

**Use Design Check Set Check Box**

Specifies whether to associate a design check set with the alignment. If this check box is cleared, then the Design Check Set list is not available.
For more information, see Alignment Design Check Sets (page 875).

**Design Check Set List**

Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

 Design Check Set Selection

Edits or copies the current design check set, or creates a design check set.

**Related procedures:**

■ Creating Offset Alignments (page 895)

# Curve and Spiral Settings Dialog Box

Use this dialog box to enter the curve settings you want to use when creating alignments or parcels using the Tangent-Tangent (With Curves) command and to specify the spiral defaults.

While you can select various combinations of Spiral In, Curve, and Spiral Out, the A value of the spirals is controlled by the radius of the curve. The length of the spirals is controlled by changing the A Value.

**NOTE** If the alignment has design criteria applied to it, the default values specified in the Curve and Spiral Settings dialog box will not be used. Instead, the minimum values in the specified design criteria file will be applied. For more information, see Creating an Alignment with the Alignment Layout Tools (page 887).

For more information on spirals and spiral formulas, see Spiral Definitions (page 939).

Type

Specifies the spiral type.

**NOTE** The spiral type you specify is used in all spiral commands on the Alignment Layout Tools toolbar.

Spiral In

Specifies whether your design includes a spiral either before a curve or before a Spiral Out.

Length

Specifies the length for the Spiral In.

A Value

Specifies the A value for the Spiral In.

Curve

Specifies how a curve is inserted: either with or without the Spiral In, with or without the Spiral Out or by itself.

Radius

Specifies a radius value. This value applies to curves and spirals. If you change the curve radius, the A value also changes.

Spiral Out

Specifies whether your design includes a Spiral Out.

Length

Specifies the length for the Spiral Out.

A Value

Specifies the A value for the Spiral Out.

# Edit Feature Settings - Alignment Dialog Box

Use this dialog box to view and change alignment-related settings.

This topic documents settings in all alignment-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the [icon] drawing icon.

- Alignment feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ⇥ alignment icon.

- Alignment command settings are identified by the ⊞ command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

These settings establish the default styles assigned to alignments and alignment-related labels.

## ⇥ Default Styles

Use these settings to specify the default styles for new alignments.

Alignment Style

  Specifies the default alignment style. Click in the Value column, and click ⋯ to select a style in the Alignment Style dialog box.

Alignment Label Set

  Specifies the default alignment label set. Click a cell in the Value column, and click ⋯ to select the label set in the Alignment Label Set dialog box.

Marker Style

  Specifies the display and appearance of points on the alignment. Click in the Value column, and click ⋯ to select a style in the Marker Style dialog box.

Line Label Style

  Specifies the default line label style. Click in the Value column, and click ⋯ to select a style in the Line Label Style dialog box.

Curve Label Style

  Specifies the default curve label style. Click in the Value column, and click ⋯ to select a style in the Curve Label Style dialog box.

Spiral Label Style

  Specifies the default spiral label style. Click in the Value column, and click ⋯ to select a style in the Spiral Label Style dialog box.

Station Offset Label Style

  Specifies the default station offset label style. Click in the Value column, and click ⋯ to select a style in the Station Offset Label Style dialog box.

Tangent Intersect Label Style

  Specifies the default tangent intersection label style. Click in the Value column, and click ⋯ to select a style in the Tangent Intersect Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### ⇥ Default Name Format

Use these settings to specify the default name formats for new alignments. Click in the Value column, and click ⬚ to make changes in the .

### ⇥ Station Indexing

Use this setting to specify the default station increment.

Default Station Index Increment
> Specifies the default value for station increments.

### ⇥ Superelevation Options

Use these settings to to specify defaults for calculating superelevation specifications.

Corridor Type
> Specifies the type of roadway as Undivided (single carriage way) or Divided (dual carriage way).

Cross Section Shape
> CrownedSpecifies the cross sectional shape of the roadway as Planar or :
>
> - Planar: There is no change in slope between the travel lanes.
>
> - Crowned: When no superelevation is applied, travel lanes slope downward in opposite directions from a common crown point.

Highside Location (For Planar Roads)
> Specifies the default Highside Location for planar roads.
>
> - Left Edge: Specifies that the high side of the roadway is on the left edge on tangent (unsuperelevated) sections when the corridor type is Undivided and the Cross Section Shape is planar.
>
> - Level: The roadway is level (0% slope) on tangent sections.
>
> - Right Edge: Specifies that the high side of the roadway is on the right edge on tangent (unsuperelevated) sections, when the corridor type is Undivided and the Cross Section Shape is planar.

Nominal Width - Pivot To Edge
> Specifies the typical width of the roadway from the superelevation pivot point to the outer edge-of-traveled way. Some superelevation attainment methods require that you specify this value to calculate the length of the superelevation transition. Enter a width in the Value column or click ⬚ and select a distance in the drawing area.

Normal Lane Slope (%)
> Specifies the % slope of the roadway lanes on tangent (unsuperelevated) tangents. Typical values are -15% to -2.5% for the slope used for roadway lanes when no superelevation is applied.

Normal Shoulder Slope (%)
> Specifies the % slope value of the roadway shoulders. This value is only required if either of the shoulder superelevation methods is not set to Default. Default value for a normal shoulder slope is -5%.

Station Rounding Option
> Specifies the number of positions for station rounding. Click the Value cell and choose one of the following from the list:
>
> - None: leaves stations as they are specified in the table
>
> - 0.1: rounds to the nearest tenth value of the station
>
> - 1: rounds up to nearest full station value

- 5: rounds up to nearest fifth meter or foot

- 10: rounds up to nearest tenth meter or foot

**NOTE** The following options for Outside Shoulder Superelevation Method and Inside Shoulder Superelevation Method are only available if the Cross Section Shape is Crowned. These options are not available for Planar roads.

Outside Shoulder Superelevation Method
Specifies the method that is applied to the outside shoulder during superelevation:

- Default Slopes: Default slopes are retained and shoulders are not superelevated.

- Match Lane Slopes: Shoulder slopes match the slopes applied to the adjacent traveled ways through out the superelevation process.

- Breakover Removal: Shoulders on the outside edge of the curve are adjusted upward to match the Normal Lane Slope (%) before the lane begins to superelevate. The high-side shoulder slope matches the high-side lane slope throughout the superelevation process, and it then rotates downward back to the Normal Shoulder Slope (%) value when the lanes are back to the unsuperelevated condition.

Inside Shoulder Superelevation Method
Specifies the method that is applied to the inside shoulder during superelevation:

- Default Slopes: Default slopes are retained and shoulders are not superelevated.

- Match Lane Slopes: Shoulder slopes match the slopes applied to the adjacent traveled ways through out the superelevation process.

- Breakover Removal: Shoulders on the outside edge of the curve are adjusted upward to match the Normal Lane Slope (%) before the lane begins to superelevate. The high-side shoulder slope matches the high-side lane slope throughout the superelevation process, and it then rotates downward back to the Normal Shoulder Slope (%) value when the lanes are back to the unsuperelevated condition.

## Criteria-Based Design Options

Use these settings to specify defaults for creating an alignment using design criteria.

Default Design Speed

Specifies the default alignment design speed. Enter a speed in the Value column or click and select a design speed in the drawing area.

Criteria-Based Design Option
Specifies whether alignments use design criteria by default. If this option is set to False, the remaining Criteria-Based Design Options are not available.

**NOTE** If the Criteria-Based Design Option is set to False, then the remaining Criteria-Based Design Options are not available.

Use Design Criteria File Option
Specifies whether to use a design criteria file by default. If this option is set to True, then the first XML file found in the *<install directory>\Data\Corridor Design Standards\<units>* directory is used by default.

Use Design Checks Option
Specifies whether to use a design check set by default.

Default Design Check Set

Specifies the default design check set. Click in the Value column, and click to select a set in the Default Design Check Set dialog box.

Design Speed Lookup Method

Specifies the table lookup rule in cases where the actual design speed does not exactly match the speed specified in the design criteria tables:

- Interpolate: Table values are interpolated to the given speed.

- Use Nearest Higher Speed: Uses the table values that correspond to the next highest design speed found in the table. (Conservative rule)

- Round To Nearest Speed: Uses the table values corresponding to the nearest speed, either higher or lower.

Radius Lookup Method

Specifies the table lookup rule in cases where the actual radius does not exactly match the speed specified in the design criteria tables:

- Interpolate: Interpolates table values for the curve group radius.

- Use Nearest Lowest Radius: Uses table values corresponding to the next lowest radius found in the table.(Conservative rule)

- Round To Nearest Radius: Uses table values corresponding to the nearest radius value found in the table.

**Related procedures:**

- Alignment Settings (page 860)

# Label Set Dialog Box

Use this dialog box to view or edit the properties of label sets, such as the label set name and included labels.

**See also:**

- Using Alignment Label Sets (page 973)

# Information Tab (Label Set Dialog Box)

Use this tab to edit primary information about a label set.

Name

Specifies the name of the label set.

Description

Specifies an optional description of the label set.

Created By

Displays the AutoCAD login name of the person who created the label set.

Date Created

Displays the date and time the label set was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the label set.

Date Modified

Displays the date and time the label set was last modified.

# Labels Tab (Label Set Dialog Box)

Use this tab to specify which labels are included in the set.

Type

Specifies the type of label to be configured and added to the label set.

**NOTE** Labels sets for superimposed profiles have only major stations and minor stations label types.

<Label Type> Label Style List

Specifies the style for the labels. Use the buttons to create or edit a label style.

 Style Selection

Specifies the style options. Create a style, copy or edit the current style selection, or pick a style from drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Add>>

Adds the specified label style to the label set.

✖

Deletes a label style from the label set.

**List of Selected Styles (Unlabeled)**

When you add a label style to a set, it is inserted into the list in the lower part of the dialog box. Use this area to define properties for the label styles and to change the selected label styles.

Type Column

Displays the label types in the current label set.

Style Column

Specifies the label style to include in the set. Click  to open the Pick Label Style dialog box.

Increment Column

Specifies the increment at which to insert major and minor station labels. Click the cell to edit the value.

**NOTE** This setting is available for station labels only.

Geometry Points To Label Column

Specifies the geometry points to label for the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types. You can label a selection of geometry points in a label type using a selected style. You can then add another instance of the same label type, which can label a second selection of geometry points using a different style. Click  to select the geometry points to label.

**Profile Label Properties**

When you add a profile label style to a label set, you can specify property values that are unique to profile label sets:

Dim Anchor Opt

Specifies the location of the anchor used to position dimension lines for certain profile view labels, such as vertical curve labels.

- **Distance Above**: Moves the dimension line anchor up the distance specified in the Dim Anchor Val field.

- **Fixed Elevation**: Places the dimension line anchor at a fixed elevation on a profile view. Specify the elevation in the Dim Anchor Val field.

- **Distance Below**: Moves the dimension line anchor down the distance specified in the Dim Anchor Val field.

Dim Anchor Val

Specifies the dimension line anchor value to apply using the Dim Anchor Opt.

Weeding

Specifies a label exclusion distance for grade break labels only. Use this setting to remove overlapping labels, making it easier to read the remaining labels. If another grade break occurs within the specified distance, it is not labeled.

**NOTE** This setting is available for Grade Break labels only.

**Related procedures:**

- [Creating a Label Set](#) (page 1547)

# Offset Parameters Dialog Box

Use this dialog box to edit the specifications of an offset alignment or widening.

Alignment Name

Identifies the alignment for which parameters are displayed.

Nominal Offset

Specifies the current offset distance for regions of the alignment that are not widened. You can edit this value.

Add a Widening button

Click to add a new widening, and then specify the parameters.

**✖** Delete Region

Click to delete the active region

**⊟▼** Expand All Categories

Click to show all levels of the parameter tree.

**⊟▲** Collapse All Categories

Click to hide all sublevels of the parameter tree, showing only the region and transition headings.

**Parameter Table**

When you click the name of a region in this table, the region is highlighted in the drawing.

Region Type

Nominal or Specific. A nominal region is generally placed before or after a widening, and the length is not critical to the widening design. A specific region is part of a widening, and has a specific length.

Start/End Station

Edit numeric values in the table, or click ![icon] and click a station value in the drawing.

Region Length

Edit the numeric value in the table, or click ![icon] and click two points in the drawing to specify a length. Changes to this value also change the end station value for the region.

Offset

Edit the numeric value in the table, or click ![icon] and click two points in the drawing to specify an offset distance.

Transition Type

Choose from one of four types:



**Linear transition**



**Curve-Line-Curve transition**



**Curve-Curve-Reverse Curve transition**



**Curve- Reverse Curve transition**

Taper Input Type

For transitions, specifies how the offset is designed: either By Length (longitudinal) or By Taper Ratio (offset: station distance).

# Profile Geometry Points Dialog Box

Use this dialog box to specify the profile geometry points that you want to label.

### Select Profile

Lists the profiles that are available for selection. Click  to select a profile from the drawing.

### Description

Displays the description of the selected profile. A profile description is set in the profile properties.

### Select Geometry Points To Label

 Select All

Selects all check boxes in the Label column.

 Deselect All

Clears all check boxes in the Label column.

Profile Geometry Points Column

Identifies the available geometric details.

**NOTE** The Abbreviations (page 1855) tab in the Drawing Settings dialog box also lists these geometry points.

Label Column

Specifies whether to label a geometry point.

### Related procedures:

■  Using Label Sets (page 973)

# Superelevation Critical Points Dialog Box

Use this dialog box to specify the superelevation critical points that you want to label.

### Select Superelevation Points To Label

 Select All

Selects all check boxes in the Label column.

 Deselect All

Clears all check boxes in the Label column.

Superelevation Points Column

Identifies the available geometric details.

**NOTE** The Abbreviations (page 1855) tab in the Drawing Settings dialog box also lists these geometry points.

Label Column

Specifies whether to label a geometry point.

**Related procedures:**

■ Using Label Sets (page 973)

# Superelevation Specification Dialog Box

Use this dialog box to edit Design Rules and Default Options for each Superelevation Region.

The settings in this dialog box are used to calculate the superelevation specifications for the alignment. The Properties pane displays a Superelevation Region for each curve group in the alignment. Each region has design rules and default options. When you click OK the dialog box closes and any existing entries on the Superelevation tab are overwritten.

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**WARNING** If the horizontal geometry violates minimum standards in the look-up tables, for example, if the radius of the curve is too small for the design speed, the Event Viewer posts a warning. It is highly recommended that you set the Show Event Viewer setting to Yes in the General section of the Edit Feature Settings - Alignment Dialog Box (page 1767) dialog box.

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### Design Rules

In this section you specify the tables to use for superelevation rate, transition length, and attainment method.

Curve Start Station

Displays the station of the beginning curve group. This value is derived from the alignment geometry.

Curve End Station

Displays the station at the end of the curve group. This value is derived from the alignment geometry.

Design Speed

Displays the design speed for the curve group. This value is derived from the Design Criteria tab of the Alignment Properties. If the curve group has two or more design speeds, the highest design speed is displayed.

Design Criteria File Name

Specifies the name of the design criteria file. This file defines minimum radius tables, superelevation attainment methods and formulae, superelevation rates and transition lengths for various roadway types and design speeds. Click in the cell to browse to the location of the design criteria file.

Superelevation Rate Table

Specifies the name of the superelevation rate table used to determine the maximum superelevation rate for the curve group. Click the arrow to display a list of the superelevation rate tables associated with the specified design criteria file.

Transition Length Table

Specifies the name of the superelevation transition length table used to calculate the transition stations for the curve group. Click the arrow to display a list of the transition length tables associated with the specified superelevation rate table.

Attainment Method

Specifies the name of the superelevation attainment method for the curve group. Click the arrow to display a list of the attainment methods associated with the specified design criteria file.

### Default Options

In this section, you specify the general settings used to calculate superelevation specifications.

**Corridor Type**

Specifies the type of roadway as Undivided (single carriage way) or Divided (dual carriage way).

**Cross Section Shape**

Specifies the cross sectional shape of the roadway as Planar or Crowned:

- Planar: There is no change in slope between the travel lanes.

- Crowned: When no superelevation is applied, travel lanes slope downward in opposite directions from a common crown point.

**Highside Location**

Specifies the default Highside Location for planar roads.

- Left Edge: Specifies that the high side of the roadway is on the left edge on tangent (unsuperelevated) sections when the corridor type is Undivided and the Cross Section Shape is planar.

- Level: The roadway is level (0% slope) on tangent sections.

- Right Edge: Specifies that the high side of the roadway is on the right edge on tangent (unsuperelevated) sections, when the corridor type is Undivided and the Cross Section Shape is planar.

**Nominal Width - Pivot To Edge**

Specifies the typical width of the roadway from the superelevation pivot point to the outer edge-of-traveled way. Some superelevation attainment methods require that you specify this value in order to calculate the length of the superelevation transition.

**Normal Lane Slope (%)**

Specifies the % slope of the roadway lanes on tangent (unsuperelevated) tangents. Typical values are -15% to -2.5%. for the slope used for roadway lanes when no superelevation is applied.

**Normal Shoulder Slope (%)**

Specifies the % slope value of the roadway shoulders. This value is only required if the Shoulder Method selection is not set to None. Default value for a normal shoulder slope is -5%.

**Station Rounding Option**

Specifies the number of positions for station rounding. Click the Value cell and choose one of the following from the list:

- None: Leaves stations as they are specified by table

- 0.1: Rounds to the nearest tenth value of the station

- 1: Rounds to nearest full station value

- 5: Rounds to nearest 5th meter or feet

- 10: Rounds to nearest 10th meter or feet

**Design Speed Lookup Method**

Specifies the superelevation table lookup rule in cases where the actual design speed does not exactly match the speed specified in the table:

- Interpolate: Table values are interpolated to the given speed.

- Use Nearest Higher Speed: Uses the table values that correspond to the next highest design speed found in the table. (Conservative rule)

- Round To Nearest Speed: Uses the table values corresponding to the nearest speed, either higher or lower.

Radius Lookup Method

Specifies the superelevation table lookup rule in cases where the actual curve group radius does not exactly match the values specified in the table:

■ Interpolate: Interpolates table values for the curve group radius.

■ Use Nearest Lowest Radius: Uses table values corresponding to the next lowest radius found in the table.(Conservative rule)

■ Round To Nearest Radius: Uses table values corresponding to the nearest radius value found in the table.

**NOTE** The following options for Outside Shoulder Superelevation Method and Inside Shoulder Superelevation Method are only available if the Cross Sections Shape is Crowned. These options are not available for Planar roads.

Outside Shoulder Superelevation Method

Specifies the method that is applied to the outside shoulder during superelevation:

■ Default Slopes: Default slopes are retained and shoulders are not superelevated.

■ Match Lane Slopes: Shoulder slopes match the slopes applied to the adjacent traveled ways through out the superelevation process.

■ Breakover Removal: Shoulders on the outside edge of the curve are adjusted upward to match the Normal Lane Slope (%) before the lane begins to superelevate. The high-side shoulder slope matches the high-side lane slope throughout the superelevation process, and it then rotates downward back to the Normal Shoulder Slope (%) value when the lanes are back to the un-superelevated condition.

Inside Shoulder Superelevation Method

Specifies the method that is applied to the inside shoulder during superelevation:

■ Default Slopes: Default slopes are retained and shoulders are not superelevated.

■ Match Lane Slopes: Shoulder slopes match the slopes applied to the adjacent traveled ways through out the superelevation process.

■ Breakover Removal: Shoulders on the outside edge of the curve are adjusted upward to match the Normal Lane Slope (%) before the lane begins to superelevate. The high-side shoulder slope matches the high-side lane slope throughout the superelevation process, and it then rotates downward back to the Normal Shoulder Slope (%) value when the lanes are back to the unsuperelevated condition.

**Related procedures:**

■ Superelevation (page 877)

# Assemblies and Subassemblies Dialog Boxes

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The following topics provide information about the Assemblies and Subassemblies dialog boxes.

## Assembly Properties Dialog Box

Use this dialog box to edit the properties of an assembly.

## Information Tab (Assembly Properties Dialog Box)

Use this tab to view or edit general information about the assembly.

Name
  Specifies the name of the current assembly.

Description
  Specifies an optional description for the current assembly.

Object Style
  Specifies the style used to display the current assembly. See Select Style Dialog Box (page 1805) or Assembly Style Dialog Box (page 1781) for more information.

Show Tooltips
  Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Construction Tab (Assembly Properties Dialog Box)

Use this tab to view the organizational structure of the subassemblies and subassembly groups comprising the assembly.

Item
  The left pane tree view displays the organizational structure of the assembly. Subassemblies are displayed in the tree view in the order in which they are processed (top to bottom). This is the same order in which each subassembly was added to the assembly. The tree view also shows how the subassembly groups are

connected to a primary baseline, or to a given offset line. You can delete, move or rename many items in this view using the right-click shortcut menu.

Input Value

When a subassembly is selected in the left pane, this pane lists information on the input parameter values that are used to calculate the size, shape, and geometry of the subassembly.

Value Name

Specifies the name of each input parameter associated with the subassembly.

Default Input Value

Specifies the input parameter value used in the processing for the selected subassembly. To edit this field, double-click in it.

Parameter Reference

Specifies that the output value from another subassembly is used as the input parameter value for the selected subassembly.

- **Use**: The value specified in the Get Value From field is used for the input parameter value instead of the default input parameter value.

- **Get Value From**: Specifies the output value from another subassembly that is used as the input parameter value for the selected subassembly.

Output Value

Lists read-only information about any output parameter values used during processing for the selected subassembly. These output values can be used as input values for other subassemblies. The actual output values are calculated only when the assembly is applied (when the corridor is created).

Value Name

Specifies the name of any output parameters for the selected subassembly.

Output Value

Specifies the value of any output parameters for the selected subassembly.

**Subassembly Help**

When a subassembly is selected in the tree view, click this button to display the Help topic describing the construction, behavior, and other details of the subassembly.

# Codes Tab (Assembly Properties Dialog Box)

Use this tab to edit the display styles for the codes associated with the selected assembly.

Code Set Style

Specifies the current code set style for the assembly. For more information, see Select Style Dialog Box (page 1805).

The properties table contains the following columns:

Name

Displays an expandable tree with a collection for each type of code: Link, Point, and Shape. Expand the collections to display the individual codes. For information about codes, see Understanding Point, Link, and Shape Codes (page 1406).

Description

Displays the description of each code.

Style

Displays the style assigned to each code.

Label Style

Displays the label style assigned to each code or <none> if no label style is assigned.

Render Material

Displays the render material assigned to each code or <none> if no render material is assigned.

Material Area Fill Style

Displays the Material Area Fill style assigned to each code or <none> if no label style is assigned.

# Assembly Style Dialog Box

Use this dialog box to control the display of the assembly in the drawing.

# Information Tab (Assembly Style Dialog Box)

Use this tab to edit the assembly style name and description and to review details, such as when the style was created and most recently used.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Marker Tab (Assembly Style Dialog Box)

Use this tab to specify the appearance of the assembly components (insertion point, baseline, baseline point, offset, and offset point) in the drawing.

Component

Specifies the assembly component for which you want to specify the appearance.

Use AutoCAD POINT For Marker

Specifies that the assembly component is displayed using the current AutoCAD point symbol, which is specified by the AutoCAD PDMODE and PDSIZE system variables.

Use Vertical LINE For Marker

Specifies that a vertical line is used for the component. Used for baseline and offset components. You can specify the length of the line using the Size field.

## Use Custom Marker

Use Custom Marker

Displays the assembly using the specified symbol.

Custom Marker Style

Specifies the symbol for displaying the marker. Click one of the five symbols to the left to use as a base symbol. Optionally, click the sixth symbol, the seventh symbol, or both, to superimpose them over the base symbol. The specified symbol combination is displayed in the Preview window.

## Use AutoCAD Block Symbol For Marker

Use AutoCAD BLOCK Symbol For Marker

Displays the assembly using a reference to the selected AutoCAD block. The block is scaled using the options specified under Size.

Blocks

Lists block definitions available in the drawing. Do one of the following:

■ Click a block name to select the block to be used for the assembly symbol. The block is displayed in the Preview window.

■ Right-click in the block list window and click Browse to select a block located in another folder.

The specified block is displayed in the Preview window.

**Marker Rotation Angle**

Marker Rotation Angle

Specifies the rotation angle for the symbol. Applies to all three symbol types (AutoCAD Point, Custom Marker, and AutoCAD Block). Enter a value or click ⬚ to specify an angle.

**Size**

Options

Specifies symbol scaling:

■ Use Drawing Scale: Determines size of the marker by multiplying the specified value by the drawing scale. Enter the scale factor.

■ Use Fixed Scale: Activates the Fixed Scale options.

■ Use Size in Absolute Units: Specifies that the marker size is an absolute value based on the displayed units. Enter the value. This is the only setting applicable for the baseline and offset line components.

■ Use Size Relative to Screen: Specifies that the size of the marker is a percentage of the drawing screen size. Enter the percentage.

Fixed Scale

Specifies independent fixed scale values when Options is set to Use Fixed Scale. Enter values for X, Y, or Z.

Orientation reference

Specifies the marker rotation angle:

■ Orient Marker To World Coordinates: Specifies that the maker rotation angle is relative to the world coordinate system.

■ Orient Marker To View: Specifies that the maker rotation angle is relative to the current AutoCAD view direction.

# Display Tab (Assembly Style Dialog Box)

Use this tab to change the display and visibility of assembly object components.

Assembly components include:

■ Insertion Point: Specifies the location in the drawing where the assembly is inserted.

■ Baseline: In layout mode, the symbol that is used to identify the assembly baseline in a drawing.

■ Baseline Point: Specifies the location that is used as the baseline location (point) for the assembly in the drawing.

■ Offset: If the assembly is associated with an offset, specifies the offset value.

■ Offset Point: If the assembly is associated with an offset, specifies the offset location in the drawing.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Assembly Style Dialog Box)

Use this tab to review all the information for the current assembly style. This information can be copied and pasted to the clipboard.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Code Set Style Dialog Box

Use this dialog box to edit the code set style.

# Information Tab (Code Set Style Dialog Box)

Use this tab to edit general information for the code set style.

Name

Specifies the code set style name.

Description

Specifies an optional description of the code set style.

# Codes Tab (Code Set Style Dialog Box)

Use this tab to view or change the style for each point, link, or shape code included in the code set.

Name

Displays an expandable tree with a collection for each type of code: Link, Point, and Shape. Expand the collections to display the individual codes. Right-click a code collection to add a style.

Description

Displays the description of each code.

Style

Displays the style assigned to each code. Click  to select a style in the Pick Style dialog box.

Label Style

Displays the label style assigned to each code or <none> if no label style is assigned. Click  to select a style in the Pick Style dialog box.

Render Material

Displays the render material assigned to each link code or <none> if no render material is assigned. Click

 to select a render material in the Select Render Material dialog box, or click  to select a render material from an object in the drawing. You can only assign render materials to link codes, not point codes or shape codes.

Material Area Fill Style

Displays the Material Area Fill style assigned to each link code or <none> if no label style is assigned. Click

 to select a style in the Pick Style dialog box. You can only assign material area fill style to link codes, not point codes or shape codes.

Feature Line Style

Displays the feature line style assigned to each point code. You can only assign feature line style to point

codes, not to link codes or shape codes. des. Click  to select a style in the Pick Style dialog box.

Import Codes

Adds codes from a selected subassembly, assembly, or corridor to this code set.

**Related procedures:**

- Understanding Point, Link, and Shape Codes (page 1406)
- Adding or Importing Codes to a Code Set Style (page 1415)

# Create Assembly Dialog Box

Use this dialog box to specify the initial parameters for the assembly, including a style and a layer.

Name

Specifies the name of the assembly.

Opens the Name Template (page 1806) dialog box, where you can modify the naming template.

Description

Specifies an optional description of this assembly.

## Assembly Style

Style List

Specifies the assembly style. Select a new style by clicking the drop-down arrow.

Style Selection

Specifies the style options. You can create a new style, copy or edit the current style, or pick a style from the drawing.

Style Detail

Opens the Style Detail dialog box where you can preview the style and creation information.

## Code Set Style

Style List

Specifies the code set style. Select a new style by clicking the drop-down arrow.

Style Selection

Specifies the style options. You can create a new style, or copy or edit the current style.

Style Detail

Opens the Style Detail dialog box where you can preview the style and creation information.

## Assembly Layer

Specifies the name of the layer on which this assembly object will be created. To change the layer, click the Object Layer icon and select a layer.

# Create Subassembly From Polyline Dialog Box

Use this dialog box to specify the initial parameters for a subassembly created from an existing polyline.

Name

Specifies the name of the subassembly.

Opens the Name Template (page 1806) dialog box, where you can modify the naming template.

Description

Specifies an optional description of this subassembly.

### Code Set Style

Style List

Specifies the current code set style. Select a new style by clicking the drop-down arrow.

Style Selection

Specifies the style options. You can create a new style, copy or edit the current style, or pick a style from the drawing.

Style Detail

Opens the Style Detail dialog box where you can preview the style and creation information.

### Subassembly Layer

Displays the name of the layer on which this subassembly object will be created. To change the layer, click the Object Layer icon and select a layer.

### Conversion Options

Mid-Ordinate Distance

If the subassembly object contains curves, all curves will be tessellated (converted into a finite number of segments) using this value to determine the length of the tessellated segments.

Link Creation

Specifies symbol scaling:

- None: Generates no links. Only points will be created at the vertices.

- Single: Creates one link from all the segments created from the object selected. This is provided to make creating shapes simpler. Using the Add Shape right-click command, you only need to perform one action.

- Multiple: Creates a new link for each segment created from the object selected. This is useful for assigning link codes to different parts of the subassembly for creating surfaces and other display characteristics.

### Erase Existing Entities

To retain existing entities, as well as the new subassembly, clear this check box.

# Edit Feature Settings - Assembly Dialog Box

Use this dialog box to view and change assembly-specific settings.

This topic documents settings in all assembly-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the  drawing icon.

- Assembly feature settings are listed near the top of this dialog box, after the General property group, and are identified by the  assembly icon.

- Assembly command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

These settings establish the default styles assigned to assembly components:

Assembly Style

Specifies the default style for assemblies. Click in the Value column. Click in the Value column, and click [...] to select a style in the Assembly Style dialog box.

Code Set Style

Specifies the default code set style. Click in the Value column, and click [...] to select a style in the Code Set Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### Default Name Format

Use these settings to specify the default name formats for assemblies, assembly offsets, and assembly groups.

Click in the Value column, and click [...] to make changes in the Name Template dialog box (page 1806).

# Edit Feature Settings - Subassembly Dialog Box

Use this dialog box to view and edit subassembly-specific settings.

This topic documents settings in all subassembly-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the drawing icon.

- Subassembly feature settings are listed near the top of this dialog box, after the General property group, and are identified by the subassembly icon.

- Subassembly command settings are identified by the command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

These settings establish the default styles assigned to subassemblies.

Code Set Style

Specifies the default code set style. Click in the Value column, and click [...] to select a style in the Code Set Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### Subassembly Name Templates

Use these settings to specify the default name formats for subassemblies created from macros and from polylines. Click in the Value column, and click ⊡ to make changes in the Name Template dialog box (page 1806).

### Convert From Polyline

**NOTE** This property group is displayed when you access the settings from the CreateSubFromPline command.

Use these settings to specify the default behavior for creating subassemblies from polylines.

Mid-ordinate Distance
Specifies mid-ordinate distance to approximate curvilinear segments. Enter a distance in the Value column or click ⊡ and select a distance in the drawing area.

Link Creation
Specifies the quantity of subassembly links (None, Single, Multiple) created from the polyline(s).

## Import Subassemblies Dialog Box

Use this dialog box to import one or more subassemblies from a package file to a tool palette or catalog.

If you have a package file that contains one or more subassemblies, you can import the subassembly or subassemblies to an AutoCAD Civil tool palette or catalog.

**Related procedures:**

■ Sharing Subassemblies (page 1403)

### Source File

Specifies the name and location of the package file (.pkt) containing one or more subassemblies.

📂

Opens the Select File dialog box where you can navigate to the package file (.pkt) containing the subassembly or subassemblies to be imported.

### Import To

You can import subassemblies to a tool palette, to a catalog, or to both locations at the same time.

Tool Palette
When this option is selected, the subassemblies will be imported to the selected tool palette. Select a tool palette by clicking the drop-down arrow. The name of the currently open tool palette is displayed (selected) by default.

Catalog Library/My Imported Tools
When this option is selected, the subassembly or subassemblies contained in the package file will be imported to the My Imported Tools catalog in the catalog library. If the My Imported Tools catalog does not exist, it will be automatically created.

## Link Style Dialog Box

This dialog box defines a link style, which controls the way a subassembly link component is displayed in a drawing.

## Information Tab (Link Style Dialog Box)

Use this tab to change the link style name and description information and to review details, such as when the style was most recently modified

For more information, see .

## Display Tab (Link Style Dialog Box)

Use this tab to change the display and visibility of the link component.

For more information, see .

## Summary Tab (Link Style Dialog Box)

Use this tab to display information about a particular link style. This information can be copied and pasted to the clipboard.

For more information, see .

# Marker Style Dialog Box

This dialog box defines a marker style, which controls the way a subassembly marker (point) component is displayed in a drawing.

## Information Tab (Marker Style Dialog Box)

Use this tab to edit the marker style name and description and to review details, such as when the style was most recently modified.

For more information, see .

## Marker Tab (Marker Style Dialog Box)

Use this tab to specify the appearance of the marker symbol in the drawing.

### Use AutoCAD Point For Marker

Use AutoCAD Point For Marker

> Displays the marker using the current AutoCAD point symbol, which is specified by the AutoCAD PDMODE and PDSIZE system variables.

### Use Custom Marker

Use Custom Marker

> Displays the marker using the specified symbol.

Custom Marker Style

> Specifies the symbol for displaying the marker. Click one of the five symbols on the left to use as a base symbol. Optionally, click the sixth symbol, the seventh symbol, or both, to superimpose them over the base symbol. The specified symbol combination is displayed in the Preview window.

### Use AutoCAD Block Symbol For Marker

Use AutoCAD Block Symbol For Marker

> Displays the marker using a reference to the selected AutoCAD block. The block is scaled using the options specified under Size.

### Blocks list

Lists block definitions available in the drawing. Do one of the following:

- Click a block name to select the block to use for the point symbol. The block is displayed in the Preview window.

- Right-click in the block list window. Click Browse to select a block located in another folder.

The specified block is displayed in the Preview window.

## Marker Rotation Angle

### Marker Rotation Angle

Specifies the rotation angle for the symbol. Applies to all three symbol types (AutoCAD Point, Custom Marker, and AutoCAD Block). Enter a value or click ![cursor] to specify an angle.

### Orientation Reference

Specifies the Marker Rotation Angle:

- World Coordinate System: Specifies that the marker rotation angle is relative to the world coordinate system.

- View: Specifies that the marker rotation angle is relative to the current AutoCAD view direction.

## Size

### Options

Specifies symbol scaling:

- Use Drawing Scale: Determines the size of the marker by multiplying the specified value by the drawing scale. Enter the scale factor.

- Use Fixed Scale: Activates the Fixed Scale commands.

- Use Size in Absolute Units: Specifies that the marker size is an absolute value based on the displayed units. Enter the value.

- Use Size Relative to Screen: The size of the marker is a percentage of the drawing screen size. Enter the percentage.

### Fixed Scale

Specifies independent fixed scale values when Options is set to Use Fixed Scale. Enter values for X, Y, or Z.

# Display Tab (Marker Style Dialog Box)

Use this tab to edit the display and visibility of the marker symbol.

For more information, see Display Tab (Style Dialog Box) (page 1801).

# Summary Tab (Marker Style Dialog Box)

Use this tab to display information about a particular marker style. This information can be copied and pasted to the clipboard.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Shape Style Dialog Box

Use this dialog box to define a shape style, which controls the way a subassembly shape component is displayed in a drawing.

## Information Tab (Shape Style Dialog Box)

Use this tab to edit the shape style name and description information and to review details, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

## Display Tab (Shape Style Dialog Box)

Use this tab to change the display and visibility of the shape's border and hatch pattern. You can also specify the hatch pattern style for this shape using this tab.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Shape Style Dialog Box)

Use this tab to display information about a particular shape style. This information can be copied and pasted to the clipboard.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Subassembly Properties Dialog Box

Use this dialog box to view or edit the properties of a subassembly in a drawing.

## Information Tab (Subassembly Properties Dialog Box)

Use this tab to view or edit the subassembly object name and description.

Name
Specifies the name of the current subassembly.

Description
Specifies an optional description of the current subassembly.

Show Tooltips
Controls whether a tooltip description is displayed for the subassembly object in the drawing.

## Parameters Tab (Subassembly Properties Dialog Box)

Use this tab to view information about subassembly input and output parameters, to edit input parameter values, and or to display the Help topic for the subassembly.

This tab also displays the method that was used to create (define) this subassembly (VBA, .NET, or from polyline).

### Input Values

Lists information on the input parameter values that are used to calculate the size, shape, and geometry of the subassembly. You can edit an input parameter value by clicking on a value in the Default Input Value column.

**NOTE** To enter values for grades, enter % values with the percent symbol included. To enter values for slopes, you can enter a ratio such as 4:1, with the colon included between two numbers, or you can enter a number that gets converted into a percent value when you click OK or Apply.

Value Name

This read-only field displays the name of each input parameter associated with the subassembly.

Default Input Value

Displays the input parameter value used in the processing for the subassembly. You can click on a value to edit it. When you edit a value and click OK or Apply, the subassembly object in the drawing is updated.

## Output Values

Lists read-only information on any output parameter values used during processing for the subassembly. These output values can be used as input values for other subassemblies. The actual output values are calculated only when the assembly is applied.

Value Name

Displays the name of any output parameters used for the subassembly.

Output Value

Displays the value for any output parameters from the subassembly.

## Defined From

This read-only field displays the method that was used to create this subassembly. If the subassembly was created using .NET, .NET is displayed. If the subassembly was created from a polyline, Entities is displayed. If the subassembly was created using .NET, .NET is displayed.

.NET Class Name

If the subassembly is created from .NET, this field displays the name of the .NET class that defines this subassembly. If the subassembly is created from VBA, or from a polyline, this field is not displayed.

.NET Assembly Name

Displays the name of the file that defines this subassembly. If the subassembly is created from VBA, or from a polyline, this field is not displayed.

Macro Name

If the subassembly is created from VBA, this field displays the name of the macro file that defines this subassembly. If the subassembly is created from a polyline, this field is not available. If the subassembly is created from .NET, this field is not displayed.

Project Name

If the subassembly is created from VBA, this field displays the name of the project that defines this subassembly. If the subassembly is created from a polyline, this field is not available. If the subassembly is created from .NET, this field is not displayed.

Subassembly Help

Clicking [...] opens the Help topic that describes the construction, behavior, and other definitions of this subassembly. If the subassembly is created from a polyline, a Help topic is not available.

# Codes Tab (Subassembly Properties Dialog Box)

Use this tab to view or edit display styles for the codes associated with the selected subassembly.

Code Set Style

Specifies the current code set style for the subassembly. For more information, see Select Style Dialog Box (page 1805).

**NOTE** When a subassembly is attached to an assembly, the code set style is governed by the assembly object's code set style, and all controls on this tab are disabled.

The properties table contains the following columns:

Name
Displays an expandable tree with a collection for each type of code: Link, Point, and Shape. Expand the collections to display the individual codes.

Description
Displays the description of each code.

Style
Displays the style assigned to each code.

Label Style
Displays the label style assigned to each code or <none> if no label style is assigned.

Render Material
Displays the render material assigned to each code or <none> if no render material is assigned.

Material Area Fill Style
Displays the Material Area Fill style assigned to each code or <none> if no label style is assigned.

**Related procedures:**

■ Understanding Point, Link, and Shape Codes (page 1406)

■

# Building Site Import Dialog Boxes

# 47

The following topics provide information about building site feature settings, the Import Building Site wizard pages, and the Building Site Style and Building Site Properties dialog boxes.

## Edit Feature Settings - Import Building Site

Use this dialog box to view and change settings that relate to building site objects.

This topic lists settings in all Edit Settings dialog boxes (drawing-level, feature-level, and command-level) that relate to building site objects.

■ Drawing-level ambient settings are identified by the  drawing icon.

■ Building site feature settings are identified by the  Building Site object icon.

■ Building site command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### Default Styles

Use these settings to specify the default styles assigned to building sites.

Building Site Style

Specifies the default building site style (page 1688). Click in the Value column, and click [...] to select a style in the Building Site Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### Default Name Format

Use this setting to specify the default name format for new building site objects.

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Building Site Name Template

Specifies the default name format for building site objects. Click in the Value column, and click  to make changes in the Name Template dialog box (page 1806).

**Related procedures:**

■ Editing a Building Site Style (page 1688)

# Import Building Site Wizard

Use this wizard to specify the settings for the imported ADSK package file.

## Select File Page (Import Building Site Wizard)

Use this page to select the ADSK package file that contains the building site model data referenced in the AutoCAD Civil drawing.

### File Information

Enter Or Browse To The File

Specifies the name and location of the ADSK package file.



Click to select the file in the Autodesk Design Package File dialog box.

Building Site Name

Specifies the name of the building site object. The name you specify becomes the name of the Building Site collection on the Prospector tab of the AutoCAD Civil Toolspace.



Click to display the Name Template dialog box. See Name Template dialog box (page 1806).

Preview

Provides a visualization of the building site object.

**Related procedures:**

■ Using the Wizard to Import a Building Site Object (page 1686)

## Confirm Units Page (Import Building Site Wizard)

Use this page to confirm drawing units referenced in the Revit architectural model.

All fields on this page are read-only. Use them for reference information.

### Building Site Units

Unit

Specifies the unit of the building site model, as it is defined in the ADSK package file.

Scale Factor

Specifies the scaling factor that is applied to Revit architectural model data during import.

### Civil Drawing Units

Drawing Units
Specifies the unit of the current Civil drawing.

Drawing Scale
Specifies the scaling factor of the current Civil drawing session.

Angular Units
Specifies the angular units of the current Civil drawing.

### Zone

Coordinate System Zone Code
Specifies the code of the coordinate system used in the Civil drawing.

Description
Specifies a description of the current zone used in the Civil drawing.

Projection
Specifies the projection method for the coordinate system.

Datum
Specifies the abbreviation for the datum on which the coordinate system is based.

**Related procedures:**

■ Using the Wizard to Import a Building Site Object (page 1686)

## Display Properties Page (Import Building Site Wizard)

Use this page to specify the display properties for the imported building site object.

### Display Properties

Building Site Layer
Displays the default layer specified in the ImportBuildingSite command settings.

Click to change the default layer in the Object Layer dialog box. See Object Layer dialog box (page 1985).

Building Site Style
Displays the default style of the building site object specified in the ImportBuildingSite command settings.

Click to modify the default style. See Editing a Building Site Style (page 1688).

**Related procedures:**

■ Using the Wizard to Import a Building Site Object (page 1686)

## Insertion Point Page (Import Building Site Wizard)

Use this page to confirm the insertion point information of the building site object in the AutoCAD Civil drawing.

**Insertion Point Information**

Specify In Drawing
  Specifies whether you wish to specify the coordinates in the drawing. Selecting this option makes the coordinate value controls unavailable.

Northing (Y)
  Specifies the northing coordinate value of the building site base point read from the ADSK package file.

Easting (X)
  Specifies the easting coordinate value of the building site base point read from the ADSK package file.

Elevation (Z)
  Specifies the elevation coordinate value of the building site base point read from the ADSK package file.

Rotation Angle
  Specifies the rotation angle of the building site base point read from the ADSK package file.

Preview
  Click to zoom in to the extents of the building site object as it would be located on the drawing.

Finish
  Click to insert the building site object at the specified coordinates. Specify In Drawing must be cleared.

**Related procedures:**

- Using the Wizard to Import a Building Site Object (page 1686)

# Building Site Style Dialog Box

Use this dialog box to specify the display properties of the building site object.

# Information Tab (Building Site Style Dialog Box)

Use this tab to specify the information related to the building site object.

Name
  Specifies the name of the style.

Created By
  Specifies the name of the person who created the style.

Date Created
  Specifies the date the style was created.

Last Modified By
  Specifies the name of the last person who modified the style.

Date Modified
  Specifies the date the style was modified.

**Related procedures:**

- Editing a Building Site Style (page 1688)

# Markers Tab (Building Site Style Dialog Box)

Use this tab to specify the style of the utility connection marker.

**Utility Connection Marker Style**

Marker Style

Specifies the marker style for the Utility Connection component of the building site object.

Click to modify the marker style in the Marker Style dialog box. See Marker Style dialog box (page 1788).

Click to view the details of the marker style in the Style Detail dialog box. See Style Detail dialog box (page 1804).

**Related procedures:**

■ Editing a Building Site Style (page 1688)

# Display Tab (Building Site Style Dialog Box)

Use this tab to modify the layer and the viewing mode for each component of the building site object.

**View Direction**

Specify one of the following views:

Plan

Displays building site components in the top-down 3D view.

Model

Displays building site components in the Model view (all but top-down view).

**Component Display**

Specify the Visibility, Layer, and Linetype properties of the following building site object components:

Building Footprint

Specifies the footprint of the architectural model.

Property Lines

Specifies the property lines that relate to the building site object.

Utilities

Specifies the utilities and utility connections that relate to the architectural model.

Site Model

Specifies the site details that relate to the architectural model, such as topography, parking, and landscaping.

Building Model

Specifies the 3D model of the building.

**Related procedures:**

■ Editing a Building Site Style (page 1688)

## Summary Tab (Building Site Style Dialog Box)

Use this tab to view a summary of the specified styles.

Information
Specifies the information associated with the building site object.

Display
Specifies the style of the building site marker.

Collapse All
Click to collapse all items in the Property column.

Expand All
Click to expand all items in the Property column.

**Related procedures:**

■ Editing a Building Site Style (page 1688)


# Building Site Properties Dialog Box

Use this dialog box to modify the name, description, and style of the building site object.

Name
Specifies the name of the building site object.

Description
Specifies any additional information about the building site object.

Building Site Style
Specifies the style of the building site object.

Click to modify the default base layer in the Object Layer dialog box.

Click to modify the default building site object style.

**Related procedures:**

■ Updating a Building Site Object Definition (page 1689)

# Common AutoCAD Civil Dialog Boxes and Tabs

# 48

Use the following links to access information about the common dialog boxes that are used by several AutoCAD Civil features.

## Edit Feature Settings - General Dialog Box

Use this dialog box to view and change general feature and command settings.

This topic documents settings in all general Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the ▭ drawing icon.

- General feature settings are listed near the top of this dialog box, after the General property group, and are identified by the 🔳 general icon.

- General command settings are identified 🔲 command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### 🔳 Default Styles

Use these settings to establish the default styles assigned to label components:

General Note Label Style

> Specifies the default style to use when inserting Note labels. Click in the Value column, and click ⋯ to select a style in the General Note Label Style dialog box.

General Line Label Style

> Specifies the default style to use when inserting Line labels. Click in the Value column, and click ⋯ to select a style in the General Line Label Style dialog box.

General Curve Label Style

Specifies the default style to use when inserting Curve labels. Click in the Value column, and click ▣ to select a style in the General Curve Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## ▣ Curve Tessellation Option

**NOTE** This property group is displayed when accessing the settings from the CreateArcByBestFit, CreateLineByBestFit, and CreateParabolaByBestFit commands.

Use these settings to specify the default behavior when curve entities are used as an input option for best fit arc, line, and parabola creation.

Tessellate Curve

Specifies whether to tesselate curve segments if they are included as an input option.

Mid-ordinate Tolerance

Specifies the mid-ordinate tolerance based on a distance across the tessellated curve segments. Enter a length in the Value column or click ▣ and select two points along a curve.

## ▣ Regression Graph Option

**NOTE** This property group is displayed when accessing the settings from the CreateArcByBestFit, CreateLineByBestFit, and CreateParabolaByBestFit commands.

Use this setting to specify the default behavior for regression analysis graphs in best fit arc, line, and parabola creation.

Spline Fit For Regression Analysis Graph

Specifies whether the regression graph displays splines on the green line that represents the best fit entity.

## ▣ Mapcheck

**NOTE** This property group is displayed when accessing the settings from the MapCheck command.

Use these settings to specify the default behavior for Mapcheck.

Use Command Line Input

Specifies whether Mapcheck will accept commands from the command line interface. Selecting No causes Mapcheck to accept input only from the Mapcheck vista.

Default Side Type

Specifies the default type of any side created using Mapcheck.

Default Curve Direction

Specifies the default direction of any curve created using Mapcheck.

Default Angle Type

Specifies the default type of any angle created using Mapcheck.

Mapcheck Color

Specifies the color for Mapcheck lines and curves which are not part of the current mapcheck. Click in the Value column, and click ▣ to select a color in the *Select Color dialog box*.

Current Mapcheck Color

Specifies the color for the current mapcheck. Click in the Value column, and click ▣ to select a color in the Select Color dialog box.

Current Point Of Beginning Color

Specifies the color of the beginning not for a selected mapcheck. Click in the Value column, and click  to select a color in the Select Color dialog box.

Current Side Color

Specifies the color for the current selected side. Click in the Value column, and click  to select a color in the Select Color dialog box.

# Styles Dialog Box

This topic describes how to use the Information, Display, and Summary tabs that are common to all object styles. Other object-specific tabs are described in the style section for each object.

# Information Tab (Style Dialog Box)

Use this tab to change the object style name and description information, and to review details, such as when the style was most recently modified.

Name

Specifies a name for the current style. This name appears on the Settings tab in Toolspace and in the style selection dialog box.

Description

Specifies an optional description for the current style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

**Related procedures:**

■ Changing Common Settings in Styles (page 56)

# Display Tab (Style Dialog Box)

Use this tab to control the visibility and display characteristics for each component that makes up an object. To change a setting, click the current value.

For Color, Linetype, Lineweight, and Plot Style, you can select specific values or you can use the reference settings of ByLayer or ByBlock. These settings are indirect and require some planning. For more information, see Using ByLayer and ByBlock to Assign Display Values (page 1803).

**NOTE** The values you enter on the Display tab take precedence over other AutoCAD Property Settings and AutoCAD Layer Settings.

## View Direction

Object styles have unique display values and varying numbers of displayed components, depending on the view direction that is selected. You can set style characteristics that control how the object components are displayed in plan view, model view, profile view, and section view. For example, you may want to set the object styles differently depending on the view direction that is selected.

Plan

Specifies how the object is displayed in plan view.

Model

Specifies how the object is displayed in model view.

Profile

Specifies how the object is displayed in a profile view.

Section

Specifies how the object is displayed in a section view.

## Component Display

Use this grid to specify the visibility and layer use for components of each object. In the Color, Linetype, and Lineweight columns you can select a specific value or set the value to BYLAYER or BYBLOCK. For more information, see Using ByLayer and ByBlock to Assign Display Values (page 1803).

Component Type

Specifies the component. The number and type of object components varies, depending on the object and whether the view is 2D or 3D.

Visible

Specifies the component visibility. Click the light bulb on or off to change the visibility.

Layer

Specifies the display layer for components, which is referenced if you select a ByLayer setting for any values. A setting of 0 means that the display layer is the same as the layer on which the component is drawn.

Color

Specifies a color for the component. Click to select a specific value or specify ByLayer or ByBlock.

Linetype

Specifies the line type for the component. You can click in the Linetype column and use the Select Linetype dialog box.
If you choose ByLayer, the linetype value is determined by examining the properties of the layer referred to in the Display tab pane.
If you choose ByBlock, the value is determined by examining the properties of the object for layer override settings first. If the object doesn't have layer overrides, the layer referred to in the Display tab pane is used to determine the properties.

LT Scale

Specifies the linetype scale for the component.

**NOTE** The Current Object Scale value controls the CELTSCALE system variable, which sets the linetype scale for new objects. The CELTSCALE value is multiplied by the LTSCALE to get the displayed linetype scale.

Lineweight

Specifies the lineweight for the component. You can click in the lineweight column and select a new lineweight in the Lineweight dialog box.

Plot Style
    Specifies the plot style for the component.

**Component Hatch Display**

Use this grid to specify the hatch patterns and properties for each object component.

Component Type
    Lists the object components that have hatch patterns.

Pattern
    Specifies the hatch pattern for the component. You can click in the column and select a new pattern from the Hatch Pattern dialog box.

Angle
    Specifies the angle value for the component.

Scale
    Specifies the scale for the hatch pattern.

**Related procedures:**

■ Layers (page 58)


# Using ByLayer and ByBlock to Assign Display Values

You can set component display properties by referencing the attributes of a drawing layer (ByLayer) or a block definition (ByBlock).

This method is less direct than setting specific values, as shown in the following example for color selection.

| If you click in the... | The component... |
| --- | --- |
| Layer column and select a specific layer | Is displayed in the color assigned to that layer. |
| Color column and select a specific color | Is displayed in that specific color. |
| Color column and select ByLayer | Is displayed in the color specified by the Layer column. |
| Color column and select ByBlock | The software determines if you made changes to the AutoCAD Layer Properties or AutoCAD Color Properties.<br>■ if you made changes, the component color reflects those changes.<br>■ if you did not make changes, the component is displayed in the color specified in the Layer column of the Display tab. |


# Summary Tab (Style Dialog Box)

This tab lists the object style properties and assigned values.

You can edit any values that appear black, but not those that are shaded (gray). To copy the Summary tab to the clipboard, right-click a cell. If you paste the contents into Excel, the column format is maintained.

Property
Displays all object style properties.

Value
Displays the value for each property.

**Related procedures:**

■ Changing Common Settings in Styles (page 56)

# Style Detail Dialog Box

Use this dialog box to preview styles that have been defined in a drawing in the AutoCAD Civil/Data/Preview folder.

After you establish styles for a particular phase of a project, you can save the drawing with those styles to the Preview folder. When you click , the Style Detail dialog box is displayed with an Information tab that contains creation information and a description, and a Preview tab that lists the drawings in the Preview folder. AutoCAD Civil has several default Preview drawings.

**NOTE** Some styles are not supported in the Style Detail Preview. Styles that are not supported display only the name and creation information.

Style List
Lists the object styles.

Preview List
Lists the defined previews in the current drawing.

**Information Tab**

Description
Displays the description of the current style.

Created By
Displays the AutoCAD login name of the person who created the style.

Date Created
Displays the date and time the style was created.

Last Modified By
Displays the AutoCAD login name of the person who last modified the style.

Date Modified
Displays the date and time the style was last modified.

**Preview Tab**

Style List
Displays a list of available styles for the object.

Preview List
Displays a list of available previews for the object.

Preview Window
Displays the selected style preview.

# Select Style Dialog Box

Use this dialog box to specify a style for an object, to create a new style, or to copy or edit an existing style.

Style List
Specifies the following options for style selection:

■ Create New

■ Copy Current Selection

■ Create Child of Current Selection (Labels only)

■ Edit Current Selection

■ Pick from Drawing



Specifies the style options. You can create a new style, copy or edit the current style selection, or pick a style from drawing.



Opens the Style Detail dialog box where you can preview the style and creation information.

**Related procedures:**

■ Object Styles (page 53)

# Select Render Material Dialog Box

Use this dialog box to specify a render material for an object.

Render Material List
Lets you specify a render material in the following ways:

■ ByBlock: Uses the render material of an associated block.

■ ByLayer: Uses the render material of the current layer that the object is on.

■ Global: Uses the Global render material.

■ Use a specified render material. You can select from the list of available render materials in the drawing.



Click to select a render material of an object in the drawing.

**Related procedures:**

■ Object Rendering (page 1577)

# Name Template Dialog Box

Use this dialog box to specify the object name when you create an object.

This name is displayed in Toolspace, and you can edit the name by changing the object properties.

### Name Formatting Template

Property Fields List

Displays the object properties that you can use as part of the name format.

Insert

Click to add the specified property to the name. The property is inserted where the cursor is placed in the Name field.

Name

Specifies the name format, which can include specified properties and text.

### Incremental Number Format

Number Style List

Specifies the number format for the specified property field.

Starting Number

Specifies the starting number for the specified property field.

Increment Value

Specifies the number that is added to each value to get the next value for the specified property field.

# Duplicate Station Selection Dialog Box

When you add a station equation to an alignment, you often create duplicate station values. Use this dialog box to specify which of the duplicate stations you want to work with.

Raw Station Back

Displays the original (raw) station values, unaffected by station equations. Select the station you want to use.

Duplicate Station

Displays the duplicate station values.

# Customize Columns Dialog Box

Use this dialog box to create and modify column display configurations.

You can create and save different views of the columns in the table from which you accessed this dialog box.

New Configuration

Displays the configuration names. The default configuration is named 'Standard'. Click New to create additional configurations.

Index

Displays the order in which the columns are displayed.

### Column

Name

Displays the name of the column.

Visibility

Specifies if the column is visible. Clear the check box to hide the column.

Width

Specifies the width of the column.

New

Click to create a new configuration. The Enter New Name dialog box is displayed from which you can enter the name of the configuration.

Delete

Click to delete the selected configuration.

---
**NOTE** You cannot delete the Standard configuration.

---

**Related procedures:**

■ Changing the Contents of a Column in a List View (page 85)

# Enter New Name Dialog Box

Use this dialog box to specify the name for a new column configuration.

The configuration details are specified in the Customize Columns dialog box.

Enter a configuration name in the field and click OK.

# Multipurpose Styles Feature Settings

Use this dialog box to specify the feature settings for Multipurpose Styles.

For more information, see Working with the Standard Settings Dialog Box Controls (page 62).

# Projection Styles Dialog Box

Use this dialog box to define styles that control the display of AutoCAD objects projected into profile views and section views.

This dialog box controls the projection style for AutoCAD objects only.

**See also:**

■ Feature Line Style Dialog Box (page 1896)

■ Point Style Dialog Box (page 2114)

■ Survey Figure Style Dialog Box (page 2390)

■ Using ByLayer and ByBlock to Assign Display Values (page 1803)

# Information Tab (Projection Style Dialog Box)

Use this tab to record basic information about the style.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Profile Tab (Projection Style Dialog Box)

Use this tab to control how this style displays AutoCAD objects in profile views.

Select object types for configuration, one at a time, and repeat as required.

Select Object

Specify the type of object to configure: AutoCAD points, solids, 3D polylines, blocks, or multi-view blocks

Object Display

Specify As Drawn or select a marker style for the current object. If a suitable marker style is not available from the list, you can create a new style and select a custom marker or AutoCAD block. The marker is displayed in the Preview pane.

**Related procedures:**

■ Marker Style Dialog Box (page 1788)

# Section Tab (Projection Style Dialog Box)

Use this tab to control how this style displays AutoCAD objects in section views.

# Display Tab (Projection Style Dialog Box)

Use this tab to configure the displayed elements of this style.

For more information, see Display Tab (Style Dialog Box) (page 1801).

# Summary Tab (Projection Style Dialog Box)

Use this tab to review and edit style settings in summary form.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Projection Properties Tab

Use this tab to set properties of projected objects in profile views and section views.

This tab appears on the dialog boxes for Profile View Properties (page 2178), and Section View Properties (page 2281) when the view includes one or more projected objects.

You can change the elevation of a projected object, modify its object style, or remove it from the view.

Name

Displays all projected objects in the current view. If you click a name, the object is highlighted in the view. Right-click a name for options to pan or zoom to the object or remove it from the view.

Style

Specifies the style of the projected object. Click a style name to change it. If you want to set a consistent style for all objects of the same type, click the main heading for the object type and select the style.

Elevation Options

Specifies how the elevation of the projected object is determined:

■ Use Object. Projected object is at the same elevation as the source object in plan view.

■ Manual. Elevation of the projected object is set manually.

- Surface. Projected object is at the elevation of the selected surface.

- Profile. Projected object is at the elevation of the selected profile.

Elevation Value

Displays the current elevation of the projected object. If the elevation option is Manual, you can change the elevation in this column.

Pick Objects

Click to select additional objects to project into the current view.

# Edit Settings Dialog Box

Use the standard Edit Settings dialog box to edit drawing ambient settings, object (feature) level settings, and command level settings.

The following describes the controls displayed in the standard Edit Settings dialog box. For a description of a specific category or setting, display the dialog box and click Help.

**Property column**

Contains settings categories and settings property names.

Category names, which display in bold, contain a list of one or more settings. Click plus (+) or minus (-) next to a category name to expand or collapse the list of settings.

**Value column**

Indicates the current value for the setting. The value is either derived from a dependency on a higher level setting or you enter the value.

**Override column**

Identifies the derivation of a setting value and whether there are any higher level dependencies. A cleared check box indicates that the value is dependent on a higher level setting. A selected check box indicates that the value has been explicitly set at the current level.

If you clear the check box, then the value you entered at the current level is reset to the value of a higher level setting.

If no check box is displayed, there is no high level dependency.

**Child Override column**

Override Indicator

Indicates that a value has been overridden at a lower level. Click to remove the lower level setting override value and replace the setting value with the current setting. The icon then changes to . When you click either OK or Apply in the dialog box, the override values are removed and replaced with the current setting.

**Lock**

Protects a settings value by assuring that any lower-level setting is fixed at the value specified at this level.

To lock a setting, click to change it to . The lock cannot be set if a child override is set.

---

**NOTE** When a setting is locked at a higher level setting, the lower-level settings display a lock to the left of the property name to indicate that it is dependent on the higher-level setting and cannot be changed.

---

**Control Icons**

 Collapse All Categories

Collapses all open categories in the dialog box.

 Expand All Categories

Expands all categories in the dialog box.

 Override All Dependencies

Selects all check boxes in the Override column and creates settings that are not dependent on upper-level settings.

When you select all Override check boxes, this option is unavailable. It is also unavailable in the highest, or the drawing level, dialog box.

**Status Update in Dialog Box**

In the lower-left corner of the dialog box, the status of overrides is displayed as you select an item in the dialog box.

**Property**

Displays the selected property name followed by a single line description of the property.

**Parent Value**

Indicates the value that the setting will have if a dependency on the higher level value is maintained.

**Parent**

Indicates the higher level setting from which this default setting is derived.

**Related procedures:**

■ Working with the Standard Settings Dialog Box Controls (page 62)

# Select Object(s) Dialog Box

Use this dialog box to select one or more objects either from a list or from the drawing area.

Object Listing

Lists the objects that are available for selection. You can select a single or multiple objects, depending on the feature you are using. To select multiple objects, hold down the SHIFT key and select the objects.



Click to select one or more objects in the drawing. The selected objects in the drawing are then displayed as selected in the object listing.

# Edit Enumeration Dialog Box

Use the arrows in this dialog box to edit the order of the enumeration values. Click in a cell to edit the current value or click in an empty cell to add an enumeration value.

# User-defined Property Classification Dialog Box

Use this dialog box to specify a name for the classification.

# New User-defined Property Dialog Box

Use this dialog box to define the properties in the classification.

Name

Specifies the name for the classification.

Description

Specifies an optional description.

Type

Specifies the property data type.

■ Boolean: Supports true or false values.

■ String: Provides a list of characters.

■ Enumeration: Provides a list of indexed values.

■ Integer: This value is not formatted.

■ Double: This value is formatted using the Unitless type.

The default values for the following data types are formatted according to the settings on the Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856) in the Drawing settings:

■ Angle

■ Area

■ Coordinate

■ Dimension

■ Direction

■ Distance

■ Elevation

■ Grade/Slope

■ Latitude

■ Longitude

■ Station

■ Volume

Lower Bound Inclusive

Specifies whether or not the lower bound value is included in the property.

Lower Bound Value

Specifies the lower bound value.

Upper Bound Inclusive

Specifies whether or not the upper bound value is included in the property.

Upper Bound Value

Specifies the upper bound value.

> **NOTE** If the property type you select is Boolean, String, or Enumeration, the lower and upper bound values are not available.

Default Value check box

Specifies whether the default property value is applied to new points when they are created.

Default Value

Specifies the default value.

Enumeration

Specifies the enumeration value by clicking in a cell and entering a new value.

**Related procedures:**

■ User-Defined Property Classifications (page 416)

# Duplicate Item Name Dialog Box

Use this dialog box to resolve situations where an item of the same name exists in the destination drawing you are attempting to copy an item to.

These items can be styles, criteria, pipe rules, and so on. The affected item is listed at the top of the dialog box.

Duplicate Item Name Resolution

Specifies how duplicate items are resolved.

■ **Overwrite**: The item in the destination drawing is overwritten with the copy.

■ **Rename**: The item in the destination drawing is renamed so it will not be overwritten. The item you are copying is copied to the drawing with its original name.

■ **Skip**: The item is not copied and the original is not overwritten.

Apply To All Duplicate Items

Specifies how the resolution is applied.

■ **Selected**: Applies the resolution to all items you are currently attempting to copy.

■ **Cleared**: Applies the resolution to the current item only. You are prompted to resolve any additional conflicts.

# Corridors Dialog Boxes

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The following topics provide information about the Corridors dialog boxes:

## Apply To A Range Of Stations Dialog Box

Use this dialog box to specify a range of stations for which to apply an assembly override.

Start Station

Specifies the first station for which to apply the assembly override.

To edit the station start, enter a value or click ⬚ to select a location in the drawing window.

End Station

Specifies the last station for which to apply the assembly override.

To edit the station start, enter a value or click ⬚ to select a location in the drawing window.

**Related procedures:**

■ Viewing and Editing Corridor Sections (page 1361)

## Corridor Boundary Definition Dialog Box

Use this dialog box to view and modify the properties of a corridor surface boundary.

Feature Line

Displays the names of the feature lines that form the boundary definition.

Baseline

Displays the baseline alignment for the feature lines.

Reverse Direction

Specifies whether the boundary is to be drawn in the reverse direction of the corridor station order.

Start Point

Specifies the start point along the corridor for the boundary. To edit the start point, enter a value or click ⬚ to select a location in the drawing.

**End Point**

Specifies the end point along the corridor for the boundary. To edit the end point, enter a value or click

 to select a location in the drawing.



Adds a new feature line to the boundary definition.



Deletes the selected feature line from the boundary definition.



Moves the selected feature line up in the boundary draw order.



Moves the selected feature line down in the boundary draw order.



Previews the boundary in the drawing as a green polyline.



Verifies that the boundary polygon is well-formed (that is, it does not cross on itself). A message is displayed indicating if the boundary is well-formed or is ill-formed with crossing edges.

**Related procedures:**

■   Adding and Editing Corridor Boundaries (page 1355)

# Corridor Parameter Override Stations Dialog Box

Use this dialog box to view and delete assembly overrides to stations in the selected region.

**Assembly Name**

Displays the name of the assembly used in the region.

**Start Station**

Displays the value of the first station in the region's range.

**End Station**

Displays the value of the last station in the region's range.



Deletes the override applied to the station selected in the list of stations.

**Delete All**

Deletes all the overrides applied to stations in the selected region and clears the list of stations.

**NOTE** Deleting one or all stations from the list only deletes the overrides applied to the stations.

The table displays a list of overridden stations and the override method.

**Station**

Displays the station's value.

**Parameter**

Displays whether or not the assembly has been edited by overriding subassembly parameters.

Geometry
   Displays whether or not a subassembly located at the station has been added, deleted, or edited by adding or deleting points or links.

**Related procedures:**

■ Editing Corridor Sections (page 1365)

■ Viewing and Deleting Overridden Stations (page 1343)

# Corridor Properties Dialog Box

Use this dialog box to change the properties of a corridor.

**See also:**

■ Managing and Editing Corridors (page 1333)

# Information Tab (Corridor Properties Dialog Box)

Use this tab to view or change general information for the corridor using the following settings:

Name
   Specifies the name of the current corridor.

Description
   Specifies an optional description for the current corridor.

Object Style
   Specifies the default corridor style used to display corridor components, such as region boundaries and assembly insertion stations. Select a style from the list or use the standard selection tools. For more information about the standard selection tools, see the Select Style (page 1805) dialog box.

Show Tooltips
   Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

**Related procedures:**

■ Creating Corridors (page 1330)

■ Editing Basic Corridor Information (page 1336)

# Parameters Tab (Corridor Properties Dialog Box)

Use this tab to view and edit the parameters of the current corridor including the baselines, regions, and controlling offsets, and setting targets.

⬚▾

   Expands all baseline categories.

⬚▴

   Collapses all baseline categories.

Turns on all baselines.



Turns off all baselines.

**Add Baseline**

Adds a baseline (alignment) to the corridor definition.

**Set All Frequencies**

Opens the Frequency To Apply Assemblies dialog box (page 1836), where you can map targets from subassembly definitions to appropriate drawing object names for the entire corridor.

**Set All Targets**

Opens the Target Mapping dialog box (page 1840), where you can map targets from subassembly definitions to appropriate drawing object names for the entire corridor. For more information, see Setting and Editing Targets (page 1343).

The parameters table contains the following columns:

**Name**

Displays a tree listing all baselines, regions, and corresponding offset alignments.

**NOTE** Offset alignments are displayed if the assembly is defined with assembly offsets. For more information, seeCreating an Assembly Offset (page 1382).

The check boxes next to the baselines or region names specify whether to show (selected) or hide (cleared) the baselines or regions. If the check box is cleared, that section of corridor is not reprocessed when underlying elements (for example, alignments, subassemblies, or surfaces) change.

**NOTE** The check boxes have a tri-state display. If only one of the regions is selected under a baseline, the check box is dimmed and unavailable.

**Alignment**

Specifies the alignment for a baseline or offset alignment. Click this field to open the Pick Horizontal Alignment dialog box where you can select an alignment.

**Profile**

Specifies the profile for a baseline offset alignment. Click this field to open the Select A Profile dialog box where you can select a profile.

**Assembly**

Specifies the assembly for the corridor region. Click to open the Select An Assembly dialog box where you can select an assembly.

**Start Station**

Specifies the start station for the corridor region or controlling offset. By default, for the first region added to a baseline, this value is set to the start station of the baseline. If one or more corridor regions already exist, this value is set to the end station of last region. Enter a value or click  to select a location in the drawing area.

**End Station**

Specifies the end station for the corridor region or controlling offset. By default, for the first region added to a baseline, this value is set to the end station of the baseline. Enter a value or click  to select a location in the drawing area.

Frequency

Specifies the frequency at which assemblies are placed for a given region. Click this field to open the Frequency To Apply Assemblies dialog box (page 1836), where you can modify the station frequency and manually add stations.

Target

Opens the Target Mapping dialog box (page 1840), where you can modify the mapping of targets from subassembly definitions to appropriate drawing object names. For more information, see Setting and Editing Targets (page 1343).

Overrides

Opens the Corridor Parameter Override Stations dialog box (page 1814), where you can view and delete assembly overrides to stations in the selected region.

Select Region From Drawing

Selects a region in the drawing to display in the parameters table.

**Related procedures:**

■ Editing Corridor Parameters (page 1336)

■ Viewing and Editing Corridor Sections (page 1361)

## Codes Tab (Corridor Properties Dialog Box)

Use this tab to view all point, link, and shape codes that are used to define the corridor.

These codes are from the subassemblies (used in assemblies) that are applied to the corridor.

---

**NOTE** To change the style, description, or label style for any of the codes displayed in this tab, edit the code set style (page 1415).

---

Code Set Style

Specifies the code set style for the corridor. Select a style or use the standard style selection tools. For information, see the Select Style dialog box (page 1805).

The properties table contains the following columns:

Name

Displays an expandable tree with a collection for each type of code: Link, Point, and Shape. Expand the collections to display the individual codes. For information about codes, see Understanding Point, Link, and Shape Codes (page 1406).

Description

Displays the description of each code.

Style

Displays the style assigned to each code.

Label Style

Displays the label style assigned to each code or <none> if no label style is assigned.

Render Material

Specifies the render material (page 1577) associated with the corridor surface.

Material Area Fill Style

Specifies the fill displayed for the material areas of the corridor surface.

**Related procedures:**

- Using Codes and Code Set Styles (page 1405)
- Editing the Code Set Style (page 1346)

## Feature Lines Tab (Corridor Properties Dialog Box)

Use this tab to change the feature line display styles or to modify how a corridor's feature lines are connected.

Code Set Style

Specifies the code set style. Select a style in the Code Set Style list or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style (page 1805) dialog box.

### Specify Feature Lines To Create

Code

Displays point codes that are used in the subassemblies for the corridor.

Description

Specifies descriptions for the corresponding point codes.

Connect

Specifies whether the point codes at each station are connected. By default, all identical point codes are connected as longitudinal feature lines. To delete feature lines between point codes, clear the check box for the applicable point code.

**NOTE** If the Connect check box is unavailable, the feature line is being used in a corridor surface definition. It cannot be disconnected. For more information, see Creating and Editing Corridor Surfaces (page 1350).

Feature Line Style

Specifies the feature line style. To change the style, click in a cell and click ⊹⊡ .

Branching

Specifies how the feature lines for the corridor are branched if a point code is used a varying number of times at different stations:

- **Inward**: The feature lines branch inward joining the innermost point codes.
- **Outward**: The feature lines branch outward, joining the outermost point codes.

Connect Extra Points

Specifies whether feature lines are joined between stations where there are a varying number of the same point codes. To connect the points, select the check box.

**Related procedures:**

- Editing Feature Lines (page 1347)

## Surfaces Tab (Corridor Properties Dialog Box)

Use this tab to create corridor surfaces.

**NOTE** Corridor surfaces use standard surface display styles.

Creates an empty corridor surface to which you can subsequently add data using the Add Data fields.



Creates individual corridor surfaces from all link codes. The corridor surfaces are listed in the grid where you can view and modify them.



Opens the Name Template (page 1806) dialog box, where you can modify the surface corridor naming template.



Deletes the surface or data component that is currently selected in the Name column of the grid.

**Add Data**

Data Type

Specifies the data type from which to create a corridor surface:

- **Feature Lines**: Creates the surface from the feature lines that connect the specified point codes.

- **Links**: Creates the surface from the specified link.

Specify Code

Specifies the available links or features lines, depending on whether the Data Type is set to Feature Lines or Links.



Adds the data specified with the Data Type and Specify Code fields to the selected surface.

---

**NOTE** You must first select a corridor surface before adding the data.

---

The properties table contains the following columns:

Name

Specifies the corridor surface name. Click ⊞ next to the surface name to display its data (link codes and feature lines).
Select or clear the check box next to the surface name to specify whether or not a dynamic surface is added to the Surfaces collection on the Toolspace Prospector tab.

Surface Style

Specifies the surface style associated with the corridor surface. Click this field to open the Pick Corridor Surface Style dialog box where you can select a style. The value is also specified on the surface properties Information tab.

Render Material

Specifies the render material  (page 1577)associated with the corridor surface. Click this field to open the Select Render Material dialog box and select the render material.

Add as Breakline

Specifies whether the corridor surface is built using breaklines (page 618).

Overhang Correction

Specifies whether an overhang is to be corrected when rendered, and whether it is to be corrected following the top links or bottom links.

Description

Specifies a description for the corridor surface.

**Related procedures:**

■ Creating and Editing Corridor Surfaces (page 1350)

## Boundaries Tab (Corridor Properties Dialog Box)

Use this tab to create boundaries on corridor surfaces.

Use boundaries to prevent triangulation outside of the daylight line of a corridor surface and prevent them from being displayed, or to render a section of the corridor surface as a material style.

**NOTE** A corridor surface must exist before you can create a boundary. For information about creating a corridor surface, see Creating and Editing Corridor Surfaces (page 1350).

The properties table contains the following columns:

Name

Specifies the corridor surface names and their corresponding boundaries. Click ⊞ next to the surface name to display its boundaries. To add a boundary to a surface, right-click the corridor surface name and click Add Interactively, Add Automatically, or Add From A Polygon.

Description

Specifies the description for the corridor surface boundary.

Render Material

Specifies the render material style (page 1577)associated with the corridor surface boundary. Click this field to open the Select Render Material dialog box and select the render material.

Definitions

Specifies the boundary definition. Click **[...]** to open the Corridor Boundary Definition (page 1813) dialog box, where you can view and modify the boundary definition.

Use Type

Specifies the boundary type:

■ **Render Only**. Renders specified areas of the corridor surface with different materials (when rendering), for example, asphalt and grass.

■ **Hide Boundary**. Creates void areas or punches holes in the corridor surface.

■ **Outside Boundary**. Defines the outer boundary of a corridor surface.

**Related procedures:**

■ Adding and Editing Corridor Boundaries (page 1355)

■ Rendering Corridor Models (page 1374)

## Slope Patterns Tab (Corridor Properties Dialog Box)

Use this tab to add slope patterns between a set of corridor feature lines.

Add Slope Pattern >>

Adds a slope pattern to the corridor. You are prompted to select the feature lines that define the slope pattern.

✖

Deletes the selected slope pattern.

The properties table contains the following columns:

Index

Displays a numerical identifier for the slope pattern.

Feature Line1

Displays the point code of the first feature line selected to define the slope pattern.

Feature Line2

Displays the point code of the second feature line selected to define the slope pattern.

Slope Pattern Style

Specifies the slope pattern style. Click ![icon] and select a style in the Pick Style dialog box or use the standard style creation tools to edit or create a style. For more information about the style creation tools, see the Select Style Dialog Box (page 1805).

Baseline

Displays the feature line's corresponding baseline.

Station Start

Specifies the start station for the slope pattern display. To edit the station start, enter a value or click ![icon] to select a location in the drawing area.

Station End

Specifies the end station for the slope pattern display. To edit the station end, enter a value or click ![icon] to select a location in the drawing area.

**Related procedures:**

■ Creating and Editing Corridor Slope Patterns (page 1359)

# Corridor Parameter Editor Vista

Use this vista to view and modify the parameters of assemblies at stations along a corridor.

**Assembly Parameters**

The assembly parameters section of the vista displays the following columns:

Name

Displays an expandable tree view of selected assemblies, subassembly groups, subassemblies, and subassembly parameters used by the selected corridor alignment.

At the top of the vista, click ![icon] to expand or ![icon] to collapse the tree view.

Design Value

Displays the default value set for the subassembly parameters.
If the default value is overridden by manually editing the subassembly in the corridor section view (for example, by grip editing subassembly parts), the values cannot be edited.

Override

Specifies subassemblies that have been edited manually in the corridor section view (for example, by grip editing subassembly parts) and the parameters that have overrides on their design values. A selected check box is displayed for any subassembly parameter that is overridden. A selected check box is displayed at the assembly level, if any of its subassemblies or parameters are overridden.

To clear an override and set the subassemblies or parameters to their design value, clear the check box. To set an override, select or enter an override value for any subassembly in the Value column. To all overrides for an assembly's subassemblies or parameters to their design values, clear the check box at the assembly level.

Value

Specifies the override value for a parameter.

When you override a value in this column, the Override column displays a check box.

**NOTE** Grayed out fields indicate that the value cannot be changed from the default.

Comment

Specifies an optional comment for overridden parameters.

For subassemblies that have been edited manually in the corridor section view (for example, by grip editing subassembly parts), a comment is added automatically to indicate that they have been edited. For subassemblies created by editing the subassembly parameters, you can edit the comment as required.

**Related procedures:**

■ Viewing and Editing Corridor Sections (page 1361)

# Corridor Section Editor Ribbon

Use this ribbon to view and modify cross-sections at stations along a corridor.

## General

Inquiry

Performs an inquiry at a selected station.

## Baselines and Offsets

Select a Baseline

Specifies the baseline alignment for a corridor for which you want to view or edit sections. Click the down arrow to select from the available alignments.

## Station Selection

Displays the corridor section view at the first or Start Station on the corridor.

Use the Start Station button and the Next Station button to view the change in daylight elevation along the corridor.

Displays the corridor section view at the Previous Station on the corridor.

Select a Station

Lists all the stations for which there are corridor section views. If you click Show Overridden Stations, this list displays only stations where assemblies were edited with override parameters. Click to select a station.

▷

Displays the corridor section view at the Next Station on the corridor.

▷|

Displays the corridor section view at the End Station on the corridor.

Use the End Station button and the Previous Station button to view the change in daylight elevation along the corridor.

**Show Overridden Stations**

Specifies that only stations with assembly overrides are displayed in the station list.

If you click this button a second time, all stations are available for display.

## Corridor Edit Tools

**Parameter Editor**

Edits the subassembly parameters of selected corridor sections.

Edit the override locations and values of selected subassemblies in the Corridor Parameter Editor vista.

**Apply to a Station Range**

Applies the subassembly overrides to a list of stations. Click to open the Apply to a Range of Stations (page 1813) dialog box, where you can specify the range.

Only the overrides that you create by editing the subassembly parameters can be applied to a range of stations. Geometric overrides, such as adding or deleting points and links, cannot be applied to a range of stations.

**Update Corridor**

Rebuilds the corridor using the subassembly overrides.

All corridor surfaces and any other data that derives from the corridor are updated with the subassembly overrides. Has no effect when no overrides are present or when the corridor is set to rebuild automatically.

**NOTE** When you close View/Edit Corridor Section Ribbon, the corridor is automatically rebuilt with the changes.

**Add Point**

Prompts you to add points to a subassembly by selecting a link in the corridor section view. The point is placed at the midpoint of the link.

The points at the end of the links retain the codes from the original link.

**Delete Point**

Prompts you to delete points from a subassembly by selecting a point in the corridor section view.

All links connected to deleted points are also deleted.

**Add Link**

Prompts you to add links by selecting points within a subassembly to connect.

You cannot add a link by connecting points from two different subassemblies.

**Delete Link**

Prompts you to delete links from a subassembly by selecting links in the corridor section view.

The points that were connected in order to create the deleted link are not deleted.

 Add Subassembly

Prompts you to add subassemblies by attaching a new subassembly to an existing subassembly or by inserting a new subassembly between two existing subassemblies.

If you insert a subassembly between two existing subassemblies, the Override column (in the assembly parameters) displays True.

 Delete Subassembly

Prompts you to delete subassemblies by selecting a subassembly to delete in the corridor section view.

If the deleted subassembly connected two subassemblies, the remaining subassemblies are attached to each other.

 Edit User Defined Code

Prompts you to edit or delete user-defined feature codes by selecting a subassembly point marker, link, or shape in the corridor section view.

If the corridor is not set to rebuild automatically, click Rebuild. Corridor surfaces and any other data that is derived from the corridor are updated using overrides.

### View Tools

 Edit/View Options

Sets the view/edit corridor section options.

In the View/Edit Corridor Section Options dialog box you can specify a new view scale, section view, or code set style, and select to automatically rebuild the corridor.

 Zoom to Extents

Click to zoom to the extents of the corridor section view.

 Zoom to an Offset and Elevation

Click to lock the zoom to elevation and offset.

 Zoom to Subassembly

Click to lock the zoom to a subassembly object. Opens the Pick Subassembly dialog box.

### Close

 Close

Closes the Corridor Section Editor ribbon and exits the corridor section view.

**Related procedures:**

■ Viewing and Editing Corridor Sections (page 1361)

# Corridor Style Dialog Box

Use this dialog box to make it easier to locate and work with corridor region boundaries and assembly insertion stations by changing their display.

The corridor style controls the appearance of corridor region boundaries, assembly insertion stations, and stations where the default properties of the subassemblies are overridden. The other elements within the corridor object such as feature lines, corridor sections, and surfaces, are controlled by their own display styles and labels.

## Information Tab (Corridor Style Dialog Box)

Use this tab to change the corridor style name and description information, and to review details about the style, such as when it was most recently modified. For more information, see Information Tab (Style Dialog Box) (page 1801).

## Display Tab (Corridor Style Dialog Box)

Use this tab to change the display and visibility of some corridor object components.

The corridor components controlled by the corridor style include:

Region Boundaries: A closed polyline entity that shows the extents of a corridor region. It follows the outermost feature line on either side of the baseline for the region, from the region start station to the region end station.

Assembly Insertion Stations: Stations where an assembly is inserted to create the corridor model geometry. A line segment that is drawn between the outermost design points on the left and right sides at the baseline alignment station where an assembly is inserted.

Parametric Override Stations: Stations where the design parameters of a subassembly have been numerically edited without changing the default values for the subassembly.

Geometric Override Stations: Stations where the design parameters of a subassembly have been manually edited using grips without changing the default values for the subassembly.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Corridor Style Dialog Box)

Use this tab to review all the information about the current corridor style.

This information can be copied and pasted to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

## Create COGO Points Dialog Box

Use this dialog box to export corridor points as COGO points.

For Entire Corridor Range
  Specifies that all points are exported for the entire length of the corridor.

For User Specified Range
  Specifies that points are exported for a certain range along the corridor. When selected, Select Baseline, Alignment Start, and Alignment End are available.

Select Baseline

Specifies the baseline alignment from which to extract the points. Select a baseline or click  to select an alignment in the drawing. Available only if For User Specified Range is selected.

Alignment Start

Specifies the start of the range along the alignment from which to extract points. To edit the start point, enter a value or click  to select a location in the drawing.

Alignment End

Specifies the end of the range along the alignment at which to extract points. To edit the end point, enter a value or click  to select a location in the drawing.

New Point Group Name

Specifies the name of the point group that will contain all the extracted points. For information about point groups, see Point Groups (page 521).

Select Point Codes To Export

Specifies the available point codes to export. Select the check box next to the point code to export the points for the specified point code. Clear the check box if you do not want to export the points.

**Related procedures:**

■ Exporting Corridor Points as COGO Points (page 1372)

# Create Corridor Dialog Box

Use this dialog box to specify the parameters for the new corridor including baselines, regions, and controlling offsets.

Corridor Name

Specifies the name of the corridor.

**NOTE** To name the corridor, click on its default name and enter a new name or use the naming template.



Opens the Name Template (page 1806) dialog box, where you can modify the surface corridor naming template.

Description

Specifies a description for the corridor.

Corridor Layer

Displays the layer that the corridor will be created on.



Opens the Object Layer dialog box where you can select a different layer for the corridor.

Corridor Style

Displays the style that is used to display corridor components, such as region boundaries and assembly insertion stations. For more information, see Corridor Styles and Display (page 1329).

Add Baseline

Adds another baseline (alignment) to the corridor definition.

Set All Targets

Opens the Target Mapping (page 1840) dialog box, where you can map targets from subassembly definitions to appropriate drawing object names for the entire corridor. If you subsequently add an assembly region that uses the same set (or subset) of targets, the mappings are automatically assigned. For more information, see Setting and Editing Targets (page 1343).

NOTE If targets are required and not set, messages are displayed in the Event Viewer. To display the Event Viewer, click General ➤ Utilities ➤ Event Viewer. For more information on the event viewer, see The Event Viewer Vista (page 1626).

The properties table contains the following columns:

Name

Displays a tree listing all baselines, regions, and corresponding offset alignments.

NOTE Offset alignments are displayed if the assembly is defined with assembly offsets. For more information, seeCreating an Assembly Offset (page 1382).

The check boxes beside the baselines and regions specify whether to show (selected) or hide (cleared) the baselines or regions. If the check box is cleared, that section of corridor is not reprocessed when underlying elements (for example, alignments, subassemblies, or surfaces) change.

NOTE The check boxes have a tri-state display. If only one of the regions is selected under a baseline, the check box is dimmed and unavailable.

Alignment

Specifies the alignment for a baseline or offset alignment. Click this field to open the Pick Horizontal Alignment dialog box where you can select an alignment.

Profile

Specifies the profile for a baseline offset alignment. Click this field to open the Select A Profile dialog box where you can select a profile.

Assembly

Opens the Select An Assembly dialog box where you can select an assembly for the corridor region.

Start Station

Specifies the start station for the region or controlling offset. By default, for the first region added to a baseline, this value is set to the start station of the baseline. If one or more corridor regions already exist, this value is set to the end station of last region. Enter a value or click ⊹ to select a location in the drawing area.

End Station

Specifies the end station for the region or controlling offset. By default, for the first region added to a baseline, this value is set to the end station of the baseline. Enter a value or click ⊹ to select a location in the drawing area.

Frequency

Specifies the frequency at which assemblies are placed for a given baseline or region. Click ⬚ to open the Frequency To Apply Assemblies dialog box (page 1836), where you can modify the station frequency and manually add stations.

Target

Click ⬚ to open the Target Mapping dialog box (page 1840), where you can map targets from subassembly definitions to appropriate drawing object names for the region. For more information, see Setting and Editing Targets (page 1343).

Overrides

This field is disabled when you are creating a corridor. For more information, see Viewing and Deleting Overridden Stations (page 1343).

**Related procedures:**

■ Creating a Corridor (page 1332)

■ Editing Corridor Parameters (page 1336)

# Create Corridor Baseline Dialog Box

Use this dialog box to create a baseline for a corridor.

Baseline Name

Specifies the format of the corridor baseline (alignment) name. Click ⬚ to open the Name Template dialog box.

Horizontal Alignment

Lists the horizontal alignments that are available. Click ⬚ to select an alignment in the drawing. The alignment selected in the drawing is then displayed as the selected alignment in the list.

**Related procedures:**

■ Adding Baselines (page 1340)

# Create Corridor Region Dialog Box

Use this dialog box to create a corridor region.

Region Name

Specifies the format of the corridor region name. Click ⬚ to open the Name Template dialog box.

Assembly Name

Lists the assemblies that are available. Click ⬚ to select an assembly in the drawing. The assembly selected in the drawing is then displayed as the selected assembly in the list.

**Related procedures:**

■ Adding and Editing Corridor Regions (page 1337)

# Create Corridor Surfaces Dialog Box

Use this dialog box to export corridor surfaces as (DTM) surface objects.

Surface

Displays the names of the corridor surfaces available for export.

Select

Specifies whether to export the corresponding surface.

Surface Style

Specifies the style for the surface. Click to open the Pick Corridor Surface Style dialog box where you can select a style.

Render Material

Specifies the render material style (page 1577) associated with the surface. Click this field to open the Select Render Material Style dialog box where you can select a style.

**Related procedures:**

■ Exporting Corridor Surfaces (page 1373)

# Create Feature Line From Corridor Dialog Box

Use this dialog box to specify feature line settings when creating a feature line from a corridor.

Site

Specifies the site in which to place the feature line. Select a site from the list or click  to select a site from the drawing.

Name

Specifies the name of the feature line. Select the check box and click  to open the Name Template dialog box.

Style

Select the check box and select a feature line style in the list. Click  to edit the style. Click  to preview the selected style. A style is optional for a feature line, but it provides useful display options and allows you to control color, visibility, and other settings.

Layer

Specifies the layer on which the feature line should be created. Click  to open the Object Layer dialog box and specify the layer. Click Use Current Layer to specify that layer be used.

Smoothing

Select this check box to enable the curve smoothing settings.

Horizontal Deviation

Specifies the maximum 2D distance within which the non-colinear points must be located from the resulting smooth curve. If set to 0, the curve will interpolate all points.

Weeding Distance

Specifies the minimum distance between co-linear points in the resulting smooth curve. The intermediate co-linear points are weeded.

Arc Inclusion Distance

Specifies the maximum distance within which the non-collinear points must be located to be converted into arcs.

Create Dynamic Link to the Corridor

Creates a dynamic link to the corridor. Then, the feature line updates dynamically with any changes to the corridor. If the feature line is created with the dynamic link, the link can be disabled with the Feature Line Properties dialog box (page 1910). When the dynamic link is disabled, it cannot be restored. However, when the feature line is not linked to the corridor, it can be edited with any of the feature line editing commands.

**Related procedures:**

- Creating Feature Lines (page 743)

- Exporting Feature Lines as Grading Feature Lines (page 1370)

## Create Simple Corridor Dialog Box

Use this dialog box to specify the initial corridor creation parameters, including corridor name, description, and layer.

Name

Specifies the name of the corridor.

> **NOTE** To name the corridor, click on its default name and enter a new name or use the naming template.

Opens the Name Template (page 1806) dialog box, where you can modify the corridor naming template.

Description

Specifies a description for the corridor.

Corridor Style

Displays the style that is used to display corridor components, such as region boundaries and assembly insertion stations. For more information, see Corridor Styles and Display (page 1329).

Corridor Layer

Displays the layer that the corridor will be created on.

Opens the Object Layer dialog box, where you can select a different layer for the corridor.

**Related procedures:**

- Creating a Simple Corridor (page 1331)

## Edit Feature Settings - Corridor Dialog Box

Use this dialog box to view and change corridor-related settings.

This topic documents settings in all corridor-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the 🗋 drawing icon.

- Corridor feature settings are listed near the top of this dialog box, after the General property group, and are identified by the 🖾 corridor icon.

- Corridor command settings are identified by the 🖾 command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856).

## Default Styles

Use these settings to establish the default styles assigned to corridor components:

Alignment Style

Specifies the default style for an alignment created from the corridor. Click in the Value column, and click to select a style in the Alignment Style dialog box.

Slope Pattern Style

Specifies the default style for the slope patterns added to corridors. Click in the Value column, and click to select a style in the Slope Pattern Style dialog box.

Code Set Style

Specifies the default code set style. Click in the Value column, and click to select a style in the Code Set Style dialog box.

Alignment Label Set

Specifies the default style for the alignment label sets created from corridors. Click in the Value column, and click to select a style in the Alignment Label Set dialog box.

Corridor Surface Style

Specifies the default corridor surface style. Click in the Value column, and click to select a style in the Corridor Surface Style dialog box.

Corridor Style

Specifies the default style for the corridor. Click in the Value column, and click to select a style in the Corridor Style dialog box.

Profile Label Set

Specifies the default profile label set style. Click in the Value column, and click to select a style in the Profile Label Set dialog box.

Profile Style

Specifies the default style for a profile created from the corridor. Click in the Value column, and click to select a style in the Profile Style dialog box.

Feature Line Style

Specifies the default style for a feature line created from a corridor. Click in the Value column, and click to select a style in the Feature Line Style dialog box.

Render Material Style

Specifies the default render material style for the corridor. Click in the Value column, and click to select a style in the Render Material Style dialog box.

For information about style selection, see .

## Default Name Format

Use these settings to specify the default name formats for corridors and corridor surfaces, as well as for

alignments and profiles from feature lines. Click in the Value column, and click to make changes in the .

## Region Highlight Graphics

Use these settings to establish the default styles assigned to corridor region:

**Highlight Baseline**

Specifies whether the baseline of a corridor region is highlighted by default. Click in the Value column, and select Yes or No.

**Baseline Color**

Specifies the color for the baseline. Click in the Value column, and click ⬚ to select a color in the *Select Color dialog box*.

**Baseline Lineweight**

Specifies the default baseline lineweight. Click in the Value column, and click ⬚ to select a style in the Lineweight dialog box.

**Highlight Region Boundary**

Specifies whether the boundary of a corridor region is highlighted by default. Click in the Value column, and select Yes or No.

**Region Boundary Color**

Specifies the color for the region boundary. Click in the Value column, and click ⬚ to select a color in the Select Color dialog box.

**Region Boundary Lineweight**

Specifies the default region boundary lineweight. Click in the Value column, and click ⬚ to select a style in the Lineweight dialog box.

**Highlight Horizontal Targets**

Specifies whether the horizontal targets of a corridor region are highlighted by default. Click in the Value column, and select Yes or No.

**Horizontal Targets Color**

Specifies the color for horizontal targets. Click in the Value column, and click ⬚ to select a color in the Select Color dialog box.

**Horizontal Targets Lineweight**

Specifies the default horizontal targets lineweight. Click in the Value column, and click ⬚ to select a style in the Lineweight dialog box.

**Highlight Internal Assemblies**

Specifies whether the internal assemblies of a corridor region are highlighted by default. Click in the Value column, and select Yes or No.

**Internal Assemblies Color**

Specifies the color for internal assemblies. Click in the Value column, and click ⬚ to select a color in the Select Color dialog box.

**Internal Assemblies Lineweight**

Specifies the default internal assemblies lineweight. Click in the Value column, and click ⬚ to select a style in the Lineweight dialog box.

## Profile Creation Option

**NOTE** This property group is displayed when you access the settings from the CreateAlignfromCorridor command.

Use this setting, to establish the defaults when exporting alignments and profiles from a corridor.

Create Profile From Corridor

Specifies whether a profile is automatically created when creating an alignment from a corridor feature line. If set to Yes, the profile is automatically added to the profile view.

## Feature Line Export Option

**NOTE** This property group is displayed when you access the settings from the CreateAlignfromCorridor or CreateGradingFeatureLine commands.

Use this setting to establish the defaults when exporting feature lines from a corridor:

Smoothen Extracted Entities

Specifies whether corridor feature lines are smoothed when exported.

## Assembly Insertion Defaults

**NOTE** This property group is displayed when you access the settings from the CreateCorridor or CreateSimpleCorridor commands.

Use these settings to establish the defaults assigned to corridor creation.

Frequency Along Tangents

Specifies the corridor station placement frequency along the tangent portion of an alignment. Enter a distance in the Value column or click  and select a distance in the drawing area.

Frequency Along Curves

Specifies the corridor station placement frequency along the curve portion of an alignment. Enter a distance in the Value column or click  and select a distance in the drawing area.

Frequency Along Spirals

Specifies the corridor station placement frequency along the spiral portion of an alignment. Enter a distance in the Value column or click  and select a distance in the drawing area.

Horizontal Geometry Points

Specifies whether corridor stations are placed at horizontal geometry points.

Superelevation Critical Points

Specifies whether corridor stations are placed at superelevation critical points (such as 'end normal crown,' 'level crown,' 'reverse crown,' and 'begin full super').

Profile Geometry Points

Specifies whether corridor stations are placed at profile geometry points.

Profile High Low Points

Specifies whether assemblies are inserted at the high and low points of the profile geometry.

Frequency Along Profile Curves

Specifies the corridor station placement frequency along curve portions of the profile geometry. Enter a distance in the Value column or click  and select a distance in the drawing area.

## Feature Line Creation

**NOTE** This property group is displayed when you access the settings from the CreateGradingFeatureLine command.

Use these settings to specify the defaults when creating feature lines.

Feature Line Name

Specifies whether a name is applied to feature lines when created.

Use Feature Line Style

Specifies whether a style is applied to feature lines when created.

Layer Setting

Specifies the layer that is applied to feature lines when created. The settings are: Use The Layer Setting (from the drawing settings); and Use The Current Layer.

Create Dynamic Link To Alignment

Specifies whether a dynamic link is created between the feature line and the alignment.

Smooth The Feature Line

Specifies whether feature lines are smoothed when exported.

Horizontal Deviation

Specifies the maximum 2D distance within which the non-collinear points must be located from the resulting smooth curve. If set to 0, the curve will interpolate all points.

Inclusion Distance

Specifies the maximum distance within which the non-collinear points must be located to be converted into arcs.

Weeding Distance

Specifies the minimum distance between co-linear points in the resulting smooth curve. The intermediate co-linear points are weeded.

## Criteria-Based Design Options

**NOTE** This property group is displayed when you access the settings from the CreateProfileFromCorridor command.

Use these settings to specify the defaults used when creating a profile using design criteria.

**NOTE** The design criteria values of the parent alignment take precedence over the default design criteria values of the profile. If the parent alignment does not use design criteria values, then the criteria-based design options in the profile are used.

Use Design Criteria File Option

Specifies whether to associate a design criteria file with the profile.
The first XML file found in the *<install directory>\Data\Corridor Design Standards\<units>* is used by default.

Use Design Checks Option

Specifies whether to associate a design check set with the profile.

Default Design Check Set

Specifies the default design check set. Click in the Value column, and click ⬚ to select a set in the Default Design Check Set dialog box.

**NOTE** This setting is not available if the Use Design Checks Option is set to False.

## View/Edit Options

**NOTE** This property group is displayed when you access the settings from the ViewEditCorridorSection command.

Use these settings to establish the default behavior for viewing corridor sections.

Default View Scale

Specifies the scale factor to use when viewing the corridor sections. For example, enter **1** to fit the section exactly to the viewing area.

NOTE The new view scale is used for the next section that is viewed.

Turn Off Unassociated Layers

Specifies whether drawing layers unassociated with the corridor are turned off or on.

Rebuild on Edit

Specifies whether the corridor model is automatically rebuilt when you edit a subassembly using the Corridor Section Editor Ribbon (page 1822).

Front Clip

Specifies the visible extent of 3D objects displayed in a corridor section, measured forward from the sampled section.

Back Clip

Specifies the visible extent of 3D objects displayed in a corridor section, measured from the sampled section back.

Station Tracker in Multiple Viewports

Specifies whether the station tracker is on.

## Grid Settings

NOTE This property group is displayed when you access the settings from the ViewEditCorridorSection command.

Use these settings to establish the default behavior for corridor section view grids:

Display Horizontal Grid

Specifies whether there are horizontal grid lines.

Display Vertical Grid

Specifies whether there are vertical grid lines.

Horizontal Grid Interval

Specifies the interval between horizontal grid lines. Enter a distance in the Value column or click and select a distance in the drawing area.

Vertical Grid Interval

Specifies the interval between vertical grid lines. Enter a distance in the Value column or click and select a distance in the drawing area.

Grid Color

Specifies the color for grid lines. Click in the Value column, and click to select a color in the *Select Color dialog box*.

Display Center Axis

Specifies whether there is a center axis line for the grid.

Center Axis Color

Specifies the color for the center axis line. Click in the Value column, and click to select a color in the Select Color dialog box.

###  Grid Text Settings

**NOTE** This property group is displayed when you access the settings from the ViewEditCorridorSection command.

Use these settings to establish the default behavior for viewing corridor section grid text:

Text Style

Specifies the text style. Click in the Value column, and click  to select a style in the Select Text Style dialog box.

Color

Specifies the color for grid text. Click in the Value column, and click  to select a style in the Select Color dialog box.

Text Height Percentage

Specifies the height of grid text as a percentage of the height of the grid.

Center Axis Annotation

Specifies whether there is annotation for the center axis line of the grid.

**Related procedures:**

■   Editing Corridor Settings (page 1333)

# Frequency to Apply Assemblies Dialog Box

Use this dialog box to add stations to a corridor.

The properties table displays the following settings:

## General

Displays general information about the corridor.

Corridor Name

Displays the corridor name.

Baseline Name

Displays the corridor baseline (alignment) name.

Current Range Start

Displays the value for the first station for the current assembly region.

Current Range End

Displays the value for the last station for the current assembly region.

## Apply Assembly

Specifies the frequency and placement settings for stations along the corridor.

**NOTE** Change the default values for the frequency and placement settings from the Edit Feature Settings dialog box for the CreateCorridor and CreateSimpleCorridor commands. For information, see Changing Corridor Creation Settings (page 1334).

Along Tangents

Specifies the assembly insertion frequency along the tangent portion of an alignment. Enter a value or click  and select a distance in the drawing.

Along Curves

Specifies the assembly insertion frequency along the curve portion of an alignment. Enter a value or click

[...] and select a distance in the drawing.

Along Spirals

Specifies the assembly insertion frequency along the spiral portion of an alignment. Enter a value or click

[...] and select a distance in the drawing.

Along Profile Curves

Specifies the assembly insertion frequency along curve portions of the profile geometry. Enter a value or

click [...] and select a distance in the drawing.

At Horizontal Geometry Points

Specifies whether the assembly should be inserted at horizontal geometry points where the horizontal alignment geometry changes (such as the start of a curve).

At Superelevation Critical Points

Specifies whether the assembly should be inserted at superelevation critical points (such as 'end normal crown,' 'level crown,' 'reverse crown,' and 'begin full super').

At Profile Geometry Points

Specifies whether the assembly should be inserted at profile geometry points where the profile geometry changes.

At Profile High/Low Points

Specifies whether the assembly should be inserted at the high and low points of the profile geometry.

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**NOTE** If both an assembly frequency setting that references horizontal geometry and a setting that references vertical geometry apply to a portion of a corridor, the setting that results in a smaller interval is used. For example, if a segment is both a horizontal tangent and a vertical curve, and the vertical curve frequency adds assemblies at more frequent intervals, the frequency specified for vertical curves is used.

---

➕

Adds a station. You are prompted to select a station location in the drawing. The station is added to the table of stations.

✖

Deletes the station selected in the table of stations.

The stations table displays the following columns:

Station

Displays the station value on the baseline.

Description

Specifies a description for the station. Click to enter a description.

**Related procedures:**

■ Adding and Editing Corridor Regions (page 1337)

■ Editing Corridor Settings (page 1333)

# Select a Feature Line Dialog Box

Use this dialog box to select a feature line when you have made an ambiguous selection in the drawing.

Feature Line

Lists the feature lines that you can select within the picked region. Click a feature line to select it.

**Related procedures:**

■  Managing and Editing Corridors (page 1333)

# Select a Profile Dialog Box

Use this dialog box to select a profile either from a list or from the drawing area.

Select An Alignment

Lists the horizontal alignments that are available.



Click to select an alignment in the drawing. The alignment selected in the drawing is then displayed as the selected alignment in the horizontal alignment listing.

Select A Profile

Displays the profiles that are available for selected alignment.



Click to select a profile in the drawing. The profile selected in the drawing is then displayed as the selected profile in the Select A Profile list.

**Related procedures:**

■  Creating Profiles (page 1046)

# Select a Subassembly to Insert

Use this dialog box to select a detached subassembly to insert when you are using the Edit/View Corridor Section Tools to override corridor and assembly parameters.sections:

Name

Lists the subassemblies in the current drawing.

Description

Specifies the optional description of the subassembly.



Click to select a subassembly in the drawing.

**Related procedures:**

■  Editing Corridor Sections (page 1365)

**Related procedures:**

■  Setting and Editing Targets (page 1343)

# Select Entities by Layer Dialog Box

Use this dialog box to select feature lines, survey figures and polylines to target either for width or offset, or for slope or elevation.

### Select feature lines, survey figures and polylines by layer

Layer Name

Displays the name of each layer with named entities.

Select

Click to select all entities on a layer for targeting.

Number Of Entities Found

Displays the number of entities found on the named layer.

**NOTE** All entities on any selected layer will be listed in Selected Entities To Target list in either the Set Width Or Offset Target or the Set Slope Or Elevation Target dialog box.

# Set Slope or Elevation Target Dialog Box

Use this dialog box to select an object to target for slope or elevation.

Select An Object Type To Target

Select whether to target Profiles or Feature Lines, Survey Figures, And 3D Polylines.

### Profiles

Select An Alignment

Select from the list any alignment whose profiles you intend to target.



Click to select an alignment in the drawing. The alignment selected in the drawing is then displayed as the selected alignment in the horizontal alignment listing.

Select Profiles

Select from the list any profile you intend to target.



Click to select a profile in the drawing. The profile selected in the drawing is then displayed as the selected profile in the Select Profiles list.

Add >>

Click to add a selected profile to the Selected Entities To Target list.

### Feature Lines, Survey Figures, And 3D Polylines

Select From Drawing

Click to select a feature line, survey or 3D polyline from the drawing.

Select By Layer

Click to select objects in the Select Entities By Layer dialog box.

Selected Entities To Target

Lists all selected target objects by Number (in order of addition to the list), Type and Name.

**NOTE** All entities on any layer selected in the Select Entities By Layer dialog box will be listed in Selected Entities To Target list.

Deletes the selected object from the target entities list.

Selection Choice If Multiple Targets Are Found

Specifies whether to select Target To Nearest Object or Target To Farthest Object, if there are two or more targets in the Selected Entities To Target list.

**Related procedures:**

■ Setting and Editing Targets (page 1343)

## Set Width or Offset Target Dialog Box

Use this dialog box to select an object to target for width or offset.

Select An Object Type To Target

Specify whether to target Alignments or Feature Lines, Survey Figures, And Polylines.

Alignments

Select from the list any alignment you intend to target and click Add >>, or click  to select an alignment from the drawing.

### Feature Lines, Survey Figures, And Polylines

Select From Drawing

Click to select a feature line, survey or polyline in the drawing.

Select By Layer

Click to select objects in the Select Entities By Layer dialog box.

Selected Entities To Target

Lists all selected target objects by Number (in order of addition to the list), Type and Name.

**NOTE** All entities on any layer selected in the Select Entities By Layer dialog box will be listed in Selected Entities To Target list.



Deletes the selected object from the target entities list.

Selection Choice If Multiple Targets Are Found

Specifies whether to select Target To Nearest Object or Target To Farthest Object, if there are two or more targets in the Selected Entities To Target list.

**Related procedures:**

■ Setting and Editing Targets (page 1343)

## Target Mapping Dialog Box

Use this dialog box to set and edit the mapping of targets in subassemblies to the names of actual objects in the drawing.

Corridor Name

Displays the name of the corridor.

Assembly Name

Displays the name of the assembly. If you are viewing the target for an entire corridor, for which there is more than one baseline, '**Varies**' is displayed.

Start Station

Displays the start station for the assembly's region or the start station on the corridor baseline. If you are viewing the target for the entire corridor for which there is more than one baseline, '**Varies**' is displayed.

End Station

Displays the end station for the assembly's region or the end station on the corridor baseline. If you are viewing the target for the entire corridor for which there is more than one baseline, '**Varies**' is displayed.

The properties table contains the following columns:

Target

Lists targets that are required by the corridor's subassemblies, categorized into three groups: Surfaces, Width Or Offset Targets, and Slope Or Elevation Targets.

Object Name

Opens either the Pick A Surface, Set Width Or Offset Target, Set Slope Or Elevation Target or Select Pipe Network dialog box, where you can select the AutoCAD Civil object name to map to the subassembly target.

**NOTE** To quickly set all surface targets to the same surface object, click <Click Here To Set All> next to the Surfaces collection and select the surface object.

Subassembly

Displays the name of the subassembly that requires the target.

Assembly Group

Displays the name of the assembly group.

**Related procedures:**

■ Setting and Editing Targets (page 1343)

# View/Edit Corridor Section Options Dialog Box

Use this dialog box to edit corridor section viewing and editing settings.

**View/Edit Options**

Code Set Style

Specifies the code set style for the corridor section view. Select a style from the list or use the standard style selection tools. For information, see the Select Style dialog box (page 1805).

Default View Scale

Specifies the scale factor to use when viewing the corridor sections. For example, enter **1** to fit the section exactly to the viewing area.

**NOTE** The new view scale is used for the subsequent section that is viewed.

Rebuild On Edit

Specifies whether the corridor model is automatically rebuilt when you edit a subassembly using the View/Edit Corridor Section Tools toolbar (page 1822).

Front Clip

Specifies the visible extent of 3D objects displayed in a corridor section, measured forward from the sampled section.

Back Clip
Specifies the visible extent of 3D objects displayed in a corridor section, measured from the sampled section back.

Station Tracker In Multiple Viewports
Specifies whether the station tracker is on.

## Grid Settings

Horizontal Grid
Specifies whether there are horizontal grid lines.

Vertical Grid
Specifies whether there are vertical grid lines.

Horizontal Grid Interval
Specifies the interval between horizontal grid lines.

Vertical Grid Interval
Specifies the interval between vertical grid lines.

Grid Color

Specifies the color for grid lines. Click  to open the Select Color dialog box.

Center Axis
Specifies whether there is a center axis line for the grid.

Center Axis Color

Specifies the color for the center axis line. Click  to open the Select Color dialog box.

## Grid Text Settings

Text Style
Specifies the text style.

Color

Specifies the color for grid text. Click  to open the Select Color dialog box.

Text Height Percentage
Specifies the height of grid text as a percentage of the height of the grid.

Center Axis Annotation
Specifies whether there is annotation for the center axis line of the grid.

**Related procedures:**

■ Viewing and Editing Corridor Sections (page 1361)

# Criteria-Based Design
# Dialog Boxes

# 50

The following topics provide information about the Criteria-Based Design dialog boxes.

## Design Criteria Editor Dialog Box

Use this dialog box to modify the design criteria file.

The design criteria file contains standards tables for design speed, superelevation attainment method, and minimum speed, radius, and length of individual alignment and profile sub-entities. If your local agency standards differ from the standards in the supplied design criteria file, you can use the Design Criteria Editor dialog box to customize the file to support your local standards.

### General Controls and Toolbar (unlabled)

New Button

Creates an empty design criteria file containing the Units, Alignments, and Profiles nodes in the design criteria hierarchy panel.

Open Button

Opens a browse dialog box, which allows you to select another design criteria file to edit.

Save Button

Saves unsaved changes to the currently open design criteria file.

Save As Button

Saves the currently open design criteria file using a new, user-specified name.

Undo Button

Reverses the most recent change to the currently open design criteria file.

Redo Button

Reverses the most recent undo operation in the currently open design criteria file.

**Make File Read-Only Check Box**

Prevents the currently open design criteria file from being modified. If selected, the Save, Save As, Undo, and Redo buttons are not available.

**Save And Close Button**

Saves unsaved changes to the currently open design criteria file and closes the Design Criteria Editor dialog box.

**Cancel Button**

Clears unsaved changes to the currently open design criteria file and closes the Design Criteria Editor dialog box.

**Help Button**

Opens the Design Criteria Editor dialog box help topic.

## Design Criteria Hierarchy Panel (Unlabeld)

The design criteria hierarchy panel displays the basic categories of information in the design criteria file.

Nodes that are preceded by ⊘ are containers for criteria tables or other container nodes. Nodes that are preceded by ⊞ are criteria tables, which contain the design criteria property values that are applied to the alignment or profile. Right-cli ⊘ node to add new ⊘ or ⊞ nodes to it. Click a ⊞ node to edit the table contents in the Attributes panel.

⊟ **Collapse All Button**

Collapses all the categories in the design criteria panel.

⊟ **Expand All Button**

Expands all the categories in the design criteria panel.

⊘ **Units Node**

Specifies the units of measure for the design criteria file. This value can be either Metric or Imperial. Select a units value, and then specify the individual unit values in the Attributes panel.

⊘ **Alignments Node**

Contains the design criteria for alignments. Select a ⊞ criteria table, and then specify the individual criteria values in the Attributes panel. The following categories and criteria are available this node:

■ Minimum Radius Tables

■ Superelevation Attainment Methods: Standard or Planar (Continuing or Opposing)

■ Superelevation Tables: Design Speed (Superelevation Design Speed or Transition Length Tables)

⊘ **Profiles Node**

Contains the design criteria for profiles. Select a ⊞ criteria table, and then specify the individual criteria values in the Attributes panel. You can create multiple Minimum K Tables in this category. Each Minimum K Table can contain any of the following criteria tables:

■ Stopping Sight Distance

■ Passing Sight Distance

■ Headlight Sight Distance

### Attributes Panel (Unlabled)

➕ o Add Button

Adds a row to the attribute table. If an existing row is selected, the new row is added below the selected row. If a row is not selected, the new row is added to the end of the attribute table.

❌ Delete Button

Deletes the currently selected rows.

Attribute Columns

Specify the attributes for the currently selected criteria table. Values in the individual cells are editable.

### Comments Panel

User-specified comments for the currently selected container or criteria table node. Comments may be added to any node in the design criteria hierarchy panel.

**Related procedures:**

- Design Criteria Files (page 867)

- Profile Standards in the Design Criteria File (page 1045)

# Design Check Editor Dialog Box

Use this dialog box to set up design checks that you can use to check your alignment or profile design against its associated design criteria.

The tables contained in the design criteria file (page 2472) contain most of the criteria used for alignment and profile design. Other criteria must be checked using mathematical expressions, which in the context of criteria-based design are known as design checks (page 2472).

Design checks are created and managed in the same manner as label expressions. To apply a design check to an alignment or profile, you must add it to a design check set (page 2472).

Design checks differ from label expressions in that they always return a true or false value. For example, a design check may be created to verify minimum tangent length. If a tangent length is greater than or equal to the minimum value defined in the design check, no notification is issued. If a tangent length is less than the minimum value defined in the design check, a warning notification is issued.

For examples of design checks, see Design Check Examples (page 871). For instructions on how to create a design check, see Creating a Design Check (page 873).

**NOTE** Do not enter functions, constants, or logical operators inside the { } brackets that surround Properties such as {Minimum Radius}.

To set up a design check, you can use the buttons in the dialog box and the keypad on your keyboard. There are menus in the dialog box from which you can choose Properties and functions to insert into the design check.

If you enter a combination of items that is invalid, a parse error appears when you click OK.

**NOTE** Because design checks return either a true or false value, they must contain one of the following logical operators: > (greater than), < (less than), >= (greater than or equals), <= (less than or equals), != (not equals), = (equals).

Name

Specifies a name for the design check.

Description
   Specifies a description that appears in the Settings tab item list view.

Design Check Box
   Displays the design check as defined from the functions and properties. You can also click in this box and enter characters such as numbers.

Calculator Buttons
   Enter numbers and logical operators (page 1848) into the Design Check box to define the expression.

Insert Property
   Inserts a property into the design check. Click the icon to display a list of properties relevant to the design check type. Select a property to insert it into the design check.

Insert Function
   Inserts a function into the expression. See functions (page 1846) for more information.

## Functions

**NOTE** Functions can be either upper case or lower case but not mixed case. The editor buttons always insert functions in upper case. Unless otherwise stated, theta is in radians.

| | |
|---|---|
| ABS(x) | Returns absolute value of x |
| ACOS(x) | Returns arccosine of x |
| ASIN(x) | Returns arcsin of x |
| ATAN(x) | Returns arctangent of x |
| ATAN2(y,x) | Returns arctangent of y/x in the correct quadrant based on signs of x and y |
| CEIL(x) | Returns the smallest integer that is greater-than or equal to x |
| COS(theta) | Returns the cosine of theta |
| COSH(theta) | Returns the hyperbolic cosine of theta |
| COT(theta) | Retruns the cotangent of theta |
| COTH(theta) | Returns the hyperbolic cotangent of theta |
| CSC(theta) | Returns the cosecant of x |
| CSCH(theta) | Returns the hyperbolic cosecant of x |
| DEG2GRD(theta) | Converts theta in degrees to gradients |
| DEG2RAD(theta) | Converts theta in degrees to radians |
| DRVSTN(x) | Returns x at the derived station along the corridor |
| EXP(x) | Returns exponential of x |

| FLOOR(x) | Returns the largest integer that is less-than or equal to x |
|---|---|
| FMOD(x,y) | Returns the floating point remainder of x/y |
| GRD2DEG(theta) | Converts theta in gradients to degrees |
| GRD2RAD(theta) | Converts theta in gradients to radians |
| IF(test,true_val,false_val) | Evaluates test - if test is non-zero evaluates and returns true_val else evaluates and returns false_val. True_val and false_val can be any expression. For example, where x is a Property: IF(x=0,1,sin(x)/x) This function tests x to see if it is zero, and if it is, the expression returns 1. If x is non-zero, the expression returns sin(x)/x. |
| LOG(x) | Returns the log (base e) of x |
| LOG10(x) | Returns the log (base 10) of x |
| MAX(a,b) | Returns maximum value of a and b |
| MIN(a,b) | Returns minimum value of a and b |
| POW(x,y) | Returns x raised to the y power |
| POW10(x) | Returns x raised to 10 |
| RAD2DEG(theta) | Converts theta in radians to degrees |
| RAD2GRD(theta) | Converts theta in radians to gradients |
| ROUND(x) | Rounds x to the nearest integer |
| SEC(theta) | Returns the secant of theta |
| SECH(theta) | Returns the hyperbolic secant of theta |
| SIN(theta) | Returns the sin of theta |
| SINH(theta) | Returns the hyperbolic sin of x |
| SQR(x) | Returns x squared (that is, x*x) |
| SQRT(x) | Returns the square root of x |
| TAN(theta) | Returns the tangent of theta |
| TANH(theta) | Returns the hyperbolic tangent of theta |
| TRUNC(x) | Truncates x to an integer value |

### Logical Operators

**NOTE** Logical operators return 1 for true and 0 for false. Since they return values, you can perform logical AND and OR with addition and multiplication. For example: ((a<b) + (b>c)) tests for a < b or b > c. ((a<b)*(b>c)) tests for a<b and b>c.

| Symbol | Operator Definition |
|--------|---------------------|
| = | logical equals |
| != | logical not equals |
| ! | logical not |
| < | logical less-than |
| > | logical greater-than |
| <= | logical less-than or equals |
| >= | logical greater-than or equals |
| + | binary addition |
| - | binary subtraction |
| * | binary multiplication |
| / | binary division |
| - | unary minus |
| + | unary plus |
| ^ | power |

**Related procedures:**

- Alignment Design Checks (page 871)

- Profile Design Checks (page 1045)

- Design Check Examples (page 871)

# Design Check Set Dialog Box

Use this dialog box to create a group of design checks for alignments or profiles.

A design check set consists of design checks for a combination of design check types. For example, you can create a design check set that consists of separate design checks for minimum and maximum A values for spirals, minimum radius for curves, and minimum spiral transition lengths.

**See also:**

- Alignment Design Check Sets (page 875)

# Information Tab (Design Check Set Dialog Box)

Use this tab to change the design check set name and description information, and to review details about the last time the design check set was modified.

Name

Specifies the name of the current design check set.

Description

Specifies the description of the current design check set.

Created By

Displays the AutoCAD login name of the person who created the design check set.

Date Created

Displays the date and time the design check set was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the design check set.

Date Modified

Displays the date and time the design check set was last modified.

# Design Checks Tab (Design Check Set Dialog Box)

Use this tab to edit the contents of an alignment or profile design check set.

Type

Specifies the type of design check to add to the design check set. The types that are available are dependent on the object for which you are creating the design check set. Alignments use Line, Curve, Spiral, and Tangent Intersection types. Profiles use Spiral, Line, and Curve types.

<Type> Checks

Specifies the user-defined design checks that are available for the design check type specified in the Type list. Select a design check from this list, and then click Add to add it to the design check set.

Label Style Control

Creates a design check, edits the current design check, or creates a copy of the current design check.

Add>> Add Label Style

Adds the design check selected in the <type> Checks list to the design check set.

Delete Design Check

Removes a design check from the design check set. Select the design check to delete by clicking in the row in the lower part of the dialog box.

## List of Selected Design Checks

When you add a design check to a set, it appears into the list in the lower part of the dialog box.

Design Check Column

Displays the names of the design checks in the current design check set.

Type Column

Displays the entity types to which the design checks apply.

Apply To Column

For profile curve design checks, specifies the type of curve that the design check will validate. Select one of the following options:

- Crest And Sag Curves

- Crest Curves Only

- Sag Curves Only

---

**NOTE** This column is not available for alignment design check sets.

---

**Related procedures:**

- Alignment Design Check Sets (page 875)

# Drawing Settings Dialog Box

# 51

The following topics provide information about the Drawing Settings dialog box.

## Units and Zone Tab (Drawing Settings Dialog Box)

Use this tab to select linear and angular units, intended plot scale, and coordinate zone for the current drawing.

Drawing Units

Specifies the linear units (feet or meters) for drawing entities in AutoCAD model space.

Angular Units

Specifies the angular units for drawing entities in AutoCAD model space:

- Degrees

- Grads

- Radians

Imperial To Metric Conversion

Specify one of the following conversion settings from the drop-down list:

- International Foot (one foot = 0.3048 Meters)

- US Survey Foot (39.37 Inches per Meter)

Scale Objects Inserted From Other Drawings

Specifies that objects inserted from another drawing will be scaled if necessary to match drawing units in the current drawing.

Set AutoCAD Variables to Match

Synchronizes AutoCAD settings with AutoCAD Civil settings. If there are no equivalent AutoCAD settings, a message appears asking if you want to match as closely as possible. The AutoCAD settings that are synchronized to the AutoCAD Civil settings include the AUNITS, DIMAUNIT, INSUNITS, and MEASUREMENT AutoCAD system variables (sysvars).

Scale

Specifies the intended plot scale in imperial or metric units.

Custom Scale

Specifies the plotted size of various annotation-related components, such as label text, ticks, and band heights. If you change the scale, all annotation objects adjust accordingly.

### Zone

NOTE If you know the code of the coordinate system you want, you can enter it directly in the Selected Coordinate System Code box. A period (.) in this box means no zone has been selected.

Categories

Specifies the geographic zone.

Available Coordinate Systems

Specifies the coordinate system within the selected zone category. If you do not select a zone category, then the default (No Datum, No Projection) is enabled. If you select a zone category but do not select a coordinate system, then, by default, the first entry in the list is selected.

Selected Coordinate System Code

Displays the coordinate system code.

Description

Displays a description of the selected zone.

Projection

Displays the projection method for the coordinate system.

Datum

Displays the abbreviation for the datum on which the coordinate system is based.

**Related procedures:**

■ Specifying Units and Zone Settings (page 63)

# Transformation Tab (Drawing Settings Dialog Box)

Use this tab to transform the coordinate system specified on the Units And Zone tab to local specifications.

Zone Description

Displays the description of the zone that is selected on the Units And Zone tab.

Apply Transform Settings

Specifies whether to apply the settings for coordinate system zone transformation. Selecting this check box enables all the other settings in the dialog box. You cannot select this check box if a zone is not specified on the Units and Zone tab.

### Apply Sea Level Scale Factor

Specifies whether to apply the settings for curvature correction to survey data so that measured horizontal distances are reduced to the distances at the mean seal level (*geodetic distances*).

Elevation

Specifies the mean elevation value of your project site with respect to the mean sea level.

Spheroid Radius

Displays the spheroid radius for the spheroid used by the coordinate system. This value is the radius of a spheroid close to the shape of the Earth at sea level, approximately 6,370 km. The value shown in this box is derived from the ellipsoid for the current zone.

## Grid Scale Factor

Computation

Specifies the type of scale factor:

- **Unity:** Uses 1.00 for all points within the zone.

- **User Defined:** Allows you to specify your own scale factor. For example, you can enter the average scale factor of the points in your survey. This value is used for all points or locations within the zone and is constant.

- **Reference Point:** Uses the scale factor of the specified reference point as the grid scale factor for all points within the zone.

- **Prismoidal Formula:** Uses the prismoidal formula to calculate the grid scale factor. This method is recommended because it accounts for the fact that every point has a different scale factor. The following equation is used to calculate prismoidal scale factor.

$$K_{eff} = \frac{K_{ref} + K_{pt} + K_{mid} \times 4}{6}$$

**Where:**

- Keff is the grid scale factor

- Kref is the scale factor of the reference point

- Kpt is the scale factor of the current point

- Kmid is the scale factor of the midpoint between the reference point and the current point.

A different value is used for each point or locations within the zone.

Scale Factor

Specifies the grid scale factor. Enabled only when you select User Defined for Computation.

## Reference Point

Use this section to specify a reference point for transformation. The reference point could be a benchmark that was used in a survey. It can be any point for which you know both the local coordinates and the grid coordinates.

⬚ Select Point In Drawing

Click to pick a reference point on the screen.

Point Number, Local Northing, Local Easting, Grid Northing, Grid Easting

Specifies a reference point by its point number, Local Northing and Easting values, or Grid Northing and Easting values.

## Rotation Point

Use this section to specify a rotation about the reference point in one of two ways: specify a rotation point, or apply a grid rotation angle.

⬚ Select Point In Drawing

Click to pick a rotation point on the screen to set the local Northing and Easting.

Point Number, Local Northing, Local Easting, Grid Northing, Grid Easting
Specifies a rotation point by its point number, Local Northing and Easting values, or Grid Northing and Easting values.

### Specify Grid Rotation Angle

Click to apply a rotation angle to grid north and azimuth instead of using a rotation point.

To North

Specifies the rotation angle to north. Enter an angle or click ⬚ and pick a point or line in the drawing. The rotation to grid north is the difference between the local coordinate system's north meridian and the grid north meridian of the current zone. If you are using True North, this value may equal the convergence angle. If you are using Magnetic North, this value is derived from the declination angle and the convergence angle.

Azimuth

Specifies the direction of the azimuth. Enter an angle or click ⬚ and pick a point or line in the drawing. Grid azimuth is the angle between the drawing north and the line defined by reference grid and rotation grid points.

### Related procedures:

- [Specifying Transformation Settings](#) (page 65)

# Object Layers Tab (Drawing Settings Dialog Box)

Use this tab to specify layers for various objects in AutoCAD Civil. For example, you can specify that you want all alignments on layer C-ROAD, all parcels on C-PROP, and so on. You can also specify prefixes and suffixes for the layer names.

If desired, label layers can be controlled through their label style. However, label object groups and single labels are individual AutoCAD entities, and they always reside on a given AutoCAD layer, as with all other objects. Use this tab to specify the default layer the labels will be on when initially created. These labels include the following:

- Note (use the General Note Label setting)

- Line & Curve (use the General Segment Label setting)

- Alignment Station Offset (use the Alignment-Labeling setting)

- Profile View Station Elevation (use the Profile View-Labeling setting)

- Surface Spot Elevation (use the TIN Surface-Labeling setting)

- Section View Offset Elevation (use the Section View-Labeling setting)

- Profile-Labeling setting

- Section-Labeling setting

- Pipe-Labeling setting

- Structure-Labeling setting

- Sample Line-Labeling setting

**NOTE** For pipes and structures, when indicating the "object name" as the modifier, use the parent network name, not the name of the pipe or structure. For the Sample Line Labeling setting, the modifier will be the Sample Line Group name, not the name of the individual sample lines.

Object

Lists all object types in AutoCAD Civil.

Layer

Displays the layer on which the object is drawn. Click to open the Layer Selection dialog box.

Modifier

Specifies whether the layer name includes a text-string modifier, and if so, the location of the modifier:

- **None:** No modifier.

- **Prefix:** Modifier is added at the beginning of the layer name.

- **Suffix:** Modifier is added at the end of the layer name.

Value

Specifies the text string to use as the modifier. You can enter an asterisk (**\***) to include the object name as the modifier in the layer name and to put each named object on its own layer. For example, if C-ROAD is the name specified for alignment layers and you add **-\*** as a suffix, an alignment named overpass is created on layer C-ROAD-OVERPASS.

Locked

Specifies whether layer settings are locked for the drawing to prevent overrides for individual objects.

Immediate And Independent Layer On/Off Control Of Display Components

Specifies whether object display is controlled by layer visibility.

- Selected: If an object layer is turned off, the visibility of object subcomponents is not affected if the subcomponents are on a different layer. For example, in a situation where an alignment is on layer C-ROAD and the tangents are on layers C-ROAD_LINE and C-ROAD_CURVE, if you turnoff the object layer C-ROAD the lines and curves will still display. However the labels will not display because the label object recognizes that the object layer is turned off.

- Cleared: If an object layer is turned off, the object subcomponents are also turned off (or not drawn) even in situations where the subcomponents are on a different layer.

**Related procedures:**

- Specifying Object Layer Settings (page 66)

# Abbreviations Tab (Drawing Settings Dialog Box)

Use this tab to control various default abbreviations used in drawing labels and reports.

For example, the default abbreviation for Spiral-Tangent Intersect is TS, meaning any spiral-tangent intersections in a drawing are labeled TS. If you need spiral-tangent intersection labels to be more detailed or different, change the abbreviation by clicking in the value column and changing the TS value.

In the Alignment Geometry Point Entity Data section the values also contain format strings that describe entity properties that will display in the label. For more information see, Property Field Modifiers (page 1509).

**NOTE** For individual abbrviation definitions, see the Glossary (page 2467).

### General Text

Used for abbreviations that are not feature specific.

Infinity

Applied to alignment spiral radius property values where infinity is returned as the value.

Left/Right

Supported as a unique label text component property field name Offset Side. The left and right side is determined by looking up-station along the entity.

### Alignment Geometry Point Text

Used for abbreviations that are specific to alignment geometry points.

Property

Specifies the geometric point or other entity.

### Alignment Geometry Point Entity Data

Used for abbreviations that are specific to alignment geometry point entities that are part of lines, curves, and spirals.

Property

Specifies the geometric point entity data.

### Superelevation

Used for abbreviations that are specific to superelevation.

Property

Specifies the superelevation specification properties.

### Profile

Used for abbreviations that are specific to profile geometry points.

Property

Specifies the geometric point or other entity.

**Related procedures:**

- Specifying Abbreviation Settings (page 67)

## Ambient Settings Tab (Drawing Settings Dialog Box)

Use this tab to specify default ambient (background) settings for a variety of settings, including units of measurement. These units and settings are used throughout AutoCAD Civil, unless they are overridden at the feature or command level.

For information about working with this dialog box, including the use of the override columns, see Working with the Standard Settings Dialog Box Controls (page 62).

### General

These settings specify a variety of features, some of which control how to display text, numbers, and units.

Plotted Unit Display Type

Specifies how to display numbers in interface controls and on the command line:

- Decimal: Display numbers with decimal points. If the drawing units are metric, this is the only option.

- Fractional: Display numbers with fractions.

Save Command Changes To Settings

Specifies whether any changes made to the settings during a command are saved for the drawing, even if the setting is marked as "locked" in the settings dialog box.

Show Event Viewer

Specifies whether the Event Viewer is displayed automatically when an event occurs.

Show Tooltips

Specifies whether to display tooltips in the drawing. This setting does not affect tooltips for toolbar icons. For example, if you move your cursor near an alignment, you see a tooltip that shows the station and offset location of the cursor. Tooltips can also show surface elevation, point, profile, and section information.

---

**NOTE** You can also control tooltip display at the feature settings level and by editing the properties of an individual object.

---

Imperial To Metric Conversion

Displays the conversion method specified on the Units and Zone tab as either US Survey Foot or International Foot.

New Entity Tool Tip State

Specifies whether to display tooltips by default for new objects.

Drawing Unit

Displays the linear units that are used in the model space of the drawing. Set this value on the Units And Zone tab of this dialog box.

Drawing Scale

Displays the intended plot scale. Set this value on the Units And Zone tab of this dialog box.

Scale Inserted Objects

Displays whether objects are scaled when they are inserted from other drawings. Set this value on the Units And Zone tab of this dialog box.

Driving Direction

Determines how curb return alignments are drawn for intersection objects. Left Side of the Road is typically selected for projects where vehicls travel on the left side of the road; for example, in countries such as the United Kingdom. In this case, curb return alignments in intersection objects are drawn starting from the left side of the outgoing road, and ending on the left side of the merging road. Right Side of the Road is typically selected for projects where vehicles travel on the right side of the road; for example, in countries such as the United States. In this case, curb return alignments in intersection objects are drawn starting from the right side of the outgoing road, and ending on the right side of the merging road. For more information, see Setting Driving Direction (page 1430).

Independent Layer On

Has the same effect as the Immediate And Independent Layer On/Off Control Of Display Components setting on the Object Layers (page 1854) tab.

**Labeling**

This setting specifies the default method for prompting for objects when inserting labels that contain Referenced Text components.

Labeling Prompt Method

- Command Line: When inserting a label that contains Referenced Text, you are prompted at the command line to select the objects to refer to in the label.

- Dialog: When inserting a label that contains Referenced Text, the Label Properties dialog box is displayed.

- Suppress Prompts: When inserting a label that contains Referenced Text, all prompting for objects is suppressed.

## Unitless

These settings specify how to display numeric values that are not specifically defined by the unit type settings in this dialog box such as distance and coordinate.

Precision

Specifies the number of digits to show to the right of the decimal.

Rounding

Specifies how numbers are rounded up or down to the number of decimal places specified in Precision:

- Round Normal: If the last displayable digit is even, and the next digit is exactly 5, the program rounds down. If the last displayable digit is odd, and the next digit is exactly 5, the program rounds up. An input value of 100.12345 with precision of 4 would be displayed as 100.1234.
  If the next digit after the last displayable digit is greater than 5, the program rounds up. If the next digit after the last displayable digit is less than 5, the program rounds down.

  **NOTE** AutoCAD is limited to 8 displayed decimal places. However, the actual precision beyond the display capability will still effect rounding. For example, 25.12500000001 would round to 25.13 with two place precision labeling.

- Round Up: Numbers are always rounded up. The value 100.12345 with precision of 4 would be 100.1235 and with precision 2 would be 100.13.

- Truncate: Numbers are limited to the specified number of decimal places without rounding. The value 100.12345 with precision of 4 would be 100.1234 and with precision 2 would be 100.12.

Sign

Specifies how to mark numbers as positive or negative:

- Sign Negative: A sign is used only with negative values. Example: 100.00, -100.00

- Bracket Negative: Brackets enclose negative values. Example: 100.00, (100.00).

- Sign Always +/-: A sign is used with both positive and negative values. Example: +100.00, -100.00

## Distance

These settings specify how to display linear distances.

Unit

Specifies the linear unit to use:

- Meter

- Kilometer

- Decimeter

- Centimeter

- Millimeter

- Foot

- Inch

- Yard

- Mile

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Dimension

These settings specify how to display linear dimensions.

Unit

Specifies the linear unit to use:

- Meter

- Kilometer

- Decimeter

- Centimeter

- Millimeter

- Foot

- Inch

- Yard

- Mile

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Coordinate

These settings specify how to display X and Y coordinates.

Unit

Specifies the linear unit to use:

- Meter

- Kilometer

- Decimeter

- Centimeter

- Millimeter

- Foot

- Inch

- Yard

- Mile

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Elevation

These settings specify how to display surface elevations.

Unit

Specifies the linear unit to use:

- Meter
- Kilometer
- Decimeter
- Centimeter
- Millimeter
- Foot
- Inch
- Yard
- Mile

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Area

These settings specify how to display surface areas.

Unit

Specifies the drawing unit to use:

- Square Meter
- Square Foot
- Acre
- Hectare
- Square Kilometer
- Square Mile
- Square Yard

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Volume

These settings specify how to display terrain volumes.

Unit

Specifies the drawing unit to use:

- Cubic Meter

- Cubic Foot

- Cubic Yard

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Speed

These settings specify how to display speeds for design criteria.

Unit

Specifies the speed as:

- Kilometer/hr.

- Meter/sec.

- Foot/sec.

- Mile/hr.

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

## Angle

These settings specify how to display deflection angles between two vectors.

Unit

Specifies the angular unit to use:

- Radian

- Degree

- Grad

**Precision**

For information, see the Unitless section of this topic.

**Rounding**

For information, see the Unitless section of this topic.

**Format**

Specifies how to display angles:

- Decimal (DD.DDDDDD)

- DD°MM'SS.SS" (no spaces)

- DD° MM' SS.SS" (spaced)

- DD.MMSSSS (decimal DMS)

**Sign**

For information, see the Unitless section of this topic.

**Drop Decimal For Whole Numbers**

Specifies whether to display whole numbers with zeros after decimals:

- Yes (example: North 90.00)

- No (example: North 90)

**Drop Leading Zeros For Degrees**

Drops leading zeros before values less than 10.

## Direction

These settings specify how to display directions.

**Unit**

Specifies the angular unit to use:

- Radian

- Degree

- Grad

**Precision**

For information, see the Unitless section of this topic.

**Rounding**

For information, see the Unitless section of this topic.

**Format**

Specifies how to display angles:

- Decimal (DD.DDDDDD)

- DD°MM'SS.SS" (no spaces)

- DD° MM' SS.SS" (spaced)

- DD.MMSSSS (decimal DMS)

Direction

Specifies how to display directions:

- Long Name (example: North 50° 45' East)

- Short Name (example: N 50° 45' E)

- Long Name Spaced

- Short Name Spaced

Capitalization

Specifies how to display direction names:

- Preserve Case

- Upper Case (example: NORTH)

- Lower Case (example: north)

- Title Caps. (example: North)

Sign

For information, see the Unitless section of this topic.

Measurement Type

Specifies how to calculate the direction of a vector:

- Bearings

- North Azimuth

- South Azimuth

Bearing Quadrant

Specifies which bearing quadrant to use:

- 1 - NE

- 2 - SE

- 3 - SW

- 4 - NW

Drop Decimal For Whole Numbers

Specifies whether to display whole numbers with zeros after decimals:

- Yes (example: North 90.00)

- No (example: North 90)

Drop Leading Zeros For Degrees

Drops leading zeros before values less than 10.

## Lat Long

These settings specify how to display latitude and longitude.

Unit

Specifies the angular unit to use:

- Radian

- Degree

■ Grad

**Precision**

For information, see the Unitless section of this topic.

**Rounding**

For information, see the Unitless section of this topic.

**Format**

Specifies how to display angles:

■ Decimal (DD.DDDDDD)

■ DD°MM'SS.SS" (no spaces)

■ DD° MM' SS.SS" (spaced)

■ DD.MMSSSS (decimal DMS)

**Direction**

Specifies the format of lat/long labels:

■ Prefix Short Name (example: N 50° 45')

■ Prefix Long Name (example: North 50° 45')

■ Suffix Short Name (example: 50° 45' N)

■ Suffix Long Name (example: 50° 45' North)

■ Signed: Includes minus signs for negative values (example: -72° 45' 35.45678")

■ Prefix Short Name Spaced

■ Prefix Long Name Spaced

■ Suffix Short Name Spaced

■ Suffix Long Name Spaced

**Capitalization**

Specifies how to display direction names:

■ Preserve Case

■ Upper Case (example: NORTH)

■ Lower Case (example: north)

■ Title Caps. (example: North)

**Sign**

For information, see the Unitless section of this topic.

**Drop Decimal For Whole Numbers**

Specifies whether to display whole numbers with zeros after decimals:

■ Yes (example: North 90.00)

■ No (example: North 90)

**Drop Leading Zeros For Degrees**

Drops leading zeros before values less than 10.

## Grade/Slope

These settings specify how to display grade and slope measurements.

Precision

  For information, see the Unitless section of this topic.

Rounding

  For information, see the Unitless section of this topic.

Format

  Specifies how to display grade and slope values:

  ■ Rise:Run (example 1:5)

  ■ Run:Rise (example 5:1)

  ■ Percent (example 20%)

  ■ Decimal (DD.DDDDDD) (example 0.20)

  ■ Per Mille (example 150‰)

Sign

  For information, see the Unitless section of this topic.

## Grade

These settings specify how to display grade measurements.

Precision

  For information, see the Unitless section of this topic.

Rounding

  For information, see the Unitless section of this topic.

Format

  Specifies how to display grade values:

  ■ Percent (example 20%)

  ■ Decimal (DD.DDDDDD) (example 0.20)

  ■ Per Mille (example 150‰)

Sign

  For information, see the Unitless section of this topic.

## Slope

These settings specify how to display slope measurements.

Precision

  For information, see the Unitless section of this topic.

Rounding

  For information, see the Unitless section of this topic.

Format

  Specifies how to display grade and slope values:

  ■ Rise:Run (example 1:5)

  ■ Run:Rise (example 5:1)

Sign

For information, see the Unitless section of this topic.

## Station

These settings specify how to display linear features that use station or chainage.

Unit

Specifies the drawing unit to use:

- Meter

- Decimeter

- Centimeter

- Millimeter

- Kilometer

- Foot

- Inch

- Yard

- Mile

Format

Specifies the station format in labels, tooltips, and elsewhere:

- Station Format: Specifies that normal station formatting is used (example: 0+010.123).

- Station Index Format: Specifies that indexing is used and applies the Station Index Increment that is specified on the Labels tab of the Alignment Properties dialog box. For example, if the Station Index Increment is 200m, then the station at 200m will appear as 1+00 and the station at 400m will appear as 2+00, and so on. To view this format in labels, you must set up the station label style to use Station Index Format (page 979) also.

- Decimal: Specifies that stations appear as decimal numbers (example: 0010.123).

Precision

For information, see the Unitless section of this topic.

Rounding

For information, see the Unitless section of this topic.

Sign

For information, see the Unitless section of this topic.

Station Delimiter Character

Specifies the separator character used to display stations:

- Plus Sign '+'

- Minus Sign '-'

- Automatic (plus or minus depending on whether the station value is positive or negative)

- Underscore '_'

- None

Station Delimiter Position

Specifies the scale to use for major stations:

- 1+0

- 1+00

- 1+000

- 1+0000

- 1+00000

Drop Decimal For Whole Numbers

Specifies whether to display station whole numbers with zeroes after decimals:

- Yes (example: 120+00)

- No (example: 120+00.000)

Drop Leading Zeros Right of Station Character

Drops leading zeros before the station character for whole number station values.

Minimum Display Width

Specifies the minimum display width for the station value, which allows leading zeros to be added if necessary. For example, assuming a minimum display width of 10 and a precision of 2, a station value of 1234.234 would be displayed as 0012+34.23.

## Transparent Commands

These settings specify the prompting behavior of transparent commands across all features. The formats used to prompt for grade and slope values are set using the Grade and Slope ambient settings.

Prompt For 3D Points

Specifies the prompt sequence (X,Y or X, Y, Z) when creating a point in 3D:

- True (prompt for 3D points)

- False (no Z prompt)

Prompt For Y before X

Specifies whether commands prompt for the Y coordinate before the X coordinate:

- True (Y first)

- False (X first)

Prompt For Easting Then Northing

Specifies whether commands prompt for the Easting before the Northing:

- True (Easting first)

- False (Northing first)

Prompt For Longitude Then Latitude

Specifies whether commands prompt for the Longitude before the Latitude:

- False (Latitude first)

- True (Longitude first)

**Related procedures:**

■ Specifying Ambient Settings (page 68)

# Google Earth Import and Export Dialog Boxes

# 52

The following topics provide information about the wizard pages for publishing Civil data files to Google Earth.

## Publish AutoCAD DWG to Google Earth Wizard

Use this wizard to export 3D design data from AutoCAD Civil to Google Earth for quick visualization of the model within the context of the Earth's surface imagery.

## Describe Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to specify information about the drawing to be published. This information is displayed in the Google Earth Places panel under My Places.

Name
    Specifies the name of the DWG model (required).

Description
    Specifies the details about the published DWG file (optional).

Hyperlink
    Specifies the url related to the description field. Only available if the description is specified.

**Related procedures:**

■ Publishing Civil Data to Google Earth (page 1669)

## Items Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to select the entities to be published. You can choose to publish either all or selected model space entities. You can also specify whether you want to publish entity information or object-specific elements, such as text or render materials.

All Model Space Entities
    Select to specify all model space entities to publish in Google Earth.

Selected Model Space Entities

Activates the specific entities selection mode. Select to specify individual model space entities to publish in Google Earth.



Click to select model space entities from the drawing to publish in Google Earth.

Publish Text

Select to include all text strings from the drawing in the published file. The text strings are displayed in the Google Earth data file as named placemarks.

**NOTE** All text elements are formatted based on Google Earth default style. Your original formatting may not be preserved.

Publish Entity Information

Select to include additional object-specific information with the published DWG file, for example, length, grade, and diameter for pipes.

Publish Materials

Select to include render materials associated with the model space entities.

**NOTE** If timespan information (page 1670)is attached to model space entities, this information is not published to Google Earth when you select to publish render materials.

**Related procedures:**

■ Publishing Civil Data to Google Earth (page 1669)


# Geo-Reference Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to transform the linear coordinates of the drawing to the geographical coordinates of the Google Earth image.

## Drawing Coordinate System Transform

If you have defined a mapping coordinate system for your drawing, select this option to automatically transform X, Y, Z coordinates of the drawing into geographical coordinates used in Google Earth. This option is not available if the drawing does not have a coordinate system defined. Use the User-Defined Transform options to perform manual coordinates transformation.

## User-Defined Transform

Select to manually transform the drawing data into the geographical coordinates of Google Earth.

Collect From Image

Click to collect the location information from the imported Google Earth image.

DWG Location

Specifies the coordinate value in the DWG linear units. Specify the following:

■ X: The X-coordinate of the point in the drawing to be associated with the specific longitudinal position on the Earth's surface.

■ Y: The Y-coordinate of the point in the drawing to be associated with the specific latitudinal position on the Earth's surface.

Click to select a point in the drawing to be associated with the specific position on the Earth's surface.

Earth Position (in Decimal Degrees)

Specifies the coordinate values of the points on the Earth's surface in decimal degrees. Specify the following:

■ Longitude: The longitudinal position of the point on the Earth's surface that is associated with the X-coordinate of a specific point on the drawing.

■ Latitude: The latitudinal position of the point on the Earth's surface that is associated with the Y-coordinate of a specific point on the drawing.



Click to open the Geographic Location dialog box. See Geographic Location Dialog Box (page 1872).

Orientation of DWG Y-axis to North

Specifies the rotation (in decimal degrees) of the positive Y-axis in the drawing to the North direction in Google Earth.



Click to open the Geographic Location dialog box. See Geographic Location Dialog Box (page 1872).

**Related procedures:**

■ Publishing Civil Data to Google Earth (page 1669)

# Nudge Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to fine-tune the location of the DWG model by offsetting its final position in Google Earth.

Offset (in DWG Linear Units)

Specifies the offset values for the DWG model in Google Earth. Specify the following:

■ North/South: Specify the amount to offset in the North/South direction.

■ East/West: Specify the amount to offset in the East/West direction.

■ Elevation: Specify the amount to offset the altitude.

Elevation Relative to Ground

Select to set the elevation values of the published entities to be relative to the ground level at the location of the DWG model in Google Earth.

Elevation Relative to Sea Level

Select to set the elevation values of the published entities to be relative to the sea level in Google Earth.

Drape Entities on Ground

Select to set the elevation values of the published entities to the ground elevation directly under the entity.

**Related procedures:**

■ Publishing Civil Data to Google Earth (page 1669)

## File Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to specify the name and location for your published file.

File

Specify the name and extension of the published data file. Google Earth supports both *.kmz* (compressed) and *.kml* (uncompressed) files.

⊡

Click to browse to the location where you want to save your DWG model data file.

**NOTE** Using the *.kml* extension may result in large data files. Use the *.kmz* extension to reduce file size.

**Related procedures:**

■  Publishing Civil Data to Google Earth (page 1669)

## Publish and View Page (Publish AutoCAD DWG to Google Earth Wizard)

Use this page to view the status of the publishing operation.

Publishing Status

Shows the publishing operation status bar.

View button

Click to view the published model in Google Earth.

**Related procedures:**

■  Publishing Civil Data to Google Earth (page 1669)

# Geographic Location Dialog Box

Use this dialog box to determine geographical coordinates of a point and specify more precisely the North direction orientation in relation to major cities around the world.

### Latitude

Specifies latitude and direction information in decimal values.

Latitude

Specifies the latitude of the current location. You can specify the value or select the location on the map. The valid range is 1 - 90.

Direction

Specifies the latitude vertical direction.

### Longitude

Specifies the longitude and direction in decimal values. Positive values represent west longitudes.

Longitude

Specifies the longitude of the current location. You can specify the value or select the location on the map. The valid range is 1 - 180.

Direction

Specifies the longitude horizontal direction.

**North Direction**

Specifies the North direction. By default, North is the positive Y direction in the world coordinate system (WCS).

Angle

Specifies the angle from 0 for the North direction.



Displays the North Direction. Move the bar to adjust the North direction angle value.

Map

Specifies a location by using the pointing device. The latitude and longitude values are updated when you select a location. If you enter latitude and longitude values, the pointer moves to show the location.

Nearest Big City

Uses the latitude and longitude values for the nearest big city that you select.

Region

Specifies the region of the world.

Nearest City

Specifies a city in the selected region.

Time Zone

Specifies the time zone. Time zones are estimated by reference to the location. You can set the time zone directly.

# Timespan for Google Earth Dialog Box

Use this dialog box to assign the temporal data to objects in your drawing to be able to display the published model data as they change over the specified period of time.

**Use Start Time**

Specifies the start time for the object display. By default, the date is set to the current date.

Date

Specifies the date when you want the object display to start.

**Use End Time**

Specifies the the end time for the object display. By default, the date is set to the current date

Date

Specifies the date when you want the object display to finish.

**Related procedures:**

■ Attaching Time Information to Model Data (page 1670)

# Grading Dialog Boxes

# 53

The following topics provide information about the Grading dialog boxes.

## Edit Feature Settings - Grading Dialog Box

Use this dialog box to view and change grading-related settings.

This topic documents settings in all grading-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the drawing icon.

- Grading feature settings are listed near the top of this dialog box, after the General property group, and are identified by the grading icon.

- Grading command settings are identifie command icon.

The precision and number format display used by all grading commands and used to display grading values is specified in the drawing ambient settings.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### Default Styles

Use these settings to specify the default styles for for new feature lines and grading groups.

Feature Line Style

Specifies the default style to apply when creating feature lines. Click in the Value column, and click to select a style in the Feature Line Style dialog box.

Grading Style

Specifies the default style to apply when creating grading. Click in the Value column, and click to select a style in the Grading Style dialog box.

Cut Slope Grading Style

Specifies the default style to apply when creating cut slope grading. Click in the Value column, and click ⋯ to select a style in the Cut Slope Grading Style dialog box.

Fill Slope Grading Style

Specifies the default style to apply when creating fill slope grading. Click in the Value column, and click ⋯ to select a style in the Fill Slope Grading Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## 🏷 Default Name Format

Use these settings to specify the default name formats for new feature lines and grading groups. Click in the Value column, and click ⋯ to make changes in the Name Template dialog box (page 1806).

## ☑ Feature Line Creation

**NOTE** This property group is displayed (in different forms) when accessing the settings from the CreateFeatureLineFromAlign, CreateFeatureLines, and DrawFeatureLine commands.

## ☑ Feature Line Creation (CreateFeatureLineFromAlign)

Use these settings to specify the default behavior for creating feature lines from alignments.

Feature Line Name

Specifies whether a name is applied to feature lines when created.

Use Feature Line Style

Specifies whether a style is applied to feature lines when created.

Layer Setting

Specifies what layer is applied to feature lines when created. The default is Use the Layer Setting which is from drawing settings.

Weed Points

Specifies whether to open the Weed Vertices dialog box so you can weed the objects when converting to a feature line. Change this value to True to open the Weed Vertices dialog box.

Create Dynamic Link

Specifies whether to create a dynamic link to the alignment. Change this value to False to prevent dynamic link creation.

Mid-ordinate Distance

Specifies the mid-ordinate distance for tessellating curves into short line segments. The default is 0.100.

Spiral Tesselation Factor

Specifies the factor for tessellating spirals into short line segments. The default is 0.25.

For more information, see Creating Feature Lines from Alignments (page 745)

## ☑ Feature Line Creation (CreateFeatureLines)

Use these settings to specify the default behavior for creating feature lines from objects.

Feature Line Name

Specifies whether a name is applied to feature lines when created.

Use Feature Line Style

Specifies whether a style is applied to feature lines when created.

Layer Setting

Specifies what layer is applied to feature lines when created. Use the layer setting (from drawing settings). Use the current layer. Use the selected entity layer.

Erase Existing Entities

Specifies whether creation commands erase selected objects when they are converted to feature lines. By default objects are erased. Change this value to False to have objects remain in the drawing.

Weed Points

Specifies whether to open the Weed Vertices dialog box so you can weed the objects when converting to a feature line. Change this value to True to open the Weed Vertices dialog box.

Assign Elevations

Specifies whether to open the Assign Elevations dialog box.

Elevation Source

Specifies whether to assign elevations from a surface, from selected gradings or to set a fixed elevation.

Include Intermediate Elevation Breaks

Specifies whether to include intermediate elevation breaks when assigning elevations from a surface or grading group.

For more information, see Creating Feature Lines from Objects (page 743)

## Feature Line Creation (DrawFeatureLine)

Use these settings to specify the default behavior for drawing feature lines.

Feature Line Name

Specifies whether a name is applied to feature lines when created.

Use Feature Line Style

Specifies whether a style is applied to feature lines when created.

Layer Setting

Specifies the default layer setting when feature lines are created.

For more information, see Drawing Feature Lines (page 748)

## Grading Creation

**NOTE** This property group is displayed when accessing the settings from the CreateGrading command.

Use this setting to specify the default behavior for creating grading.

Transition Region Length

Specifies the default length for transition regions. Enter a length in the Value column or click 🖼️ and select a distance in the drawing area. Creation commands prompt for the transition length if you select the start point of a grading to be within or near to an existing grading.

For more information, see Creating Grading from a Footprint (page 791)

## Grading Group Creation

**NOTE** This property group is displayed when accessing the settings from the CreateGradingGroup command.

Use these settings to specify the default behavior for creating grading groups.

Grading Surface Creation

Specifies whether a grading group generates an automatic surface by default.

Use Group Name

Specifies whether the automatic surface is named after the grading group by default.

Surface Style

Specifies the default style for the grading group. Click in the Value column, and click ⋯ to select a style in the Surface Style dialog box.

Surface Tessellation Increment

Specifies the default maximum distance along the feature line to add supplemental surface triangle lines

for better definition of the grading group surface. Enter a length in the Value column or click 🔲 and select a length in the drawing area.

Surface Cone Tessellation Angle

Specifies the default maximum angle between supplemental projection lines on exterior corners. Enter

an angle in the Value column or click 🔲 and select an angle in the drawing area.

**Tessellation settings control the placement of breaklines along rounded exterior corners and straight grading segments**



For more information, see Creating a New Grading Group (page 741)

## Feature Line Fit Curve

**NOTE** This property group is displayed when accessing the settings from the FitCurveFeature command.

Use these settings to specify the default behavior for creating feature lines along arcs.

Tolerance

Specifies the maximum distance from a PI along the feature to the arc that is to be inserted. When an arc is inserted, it should pass through all the feature line points (between the start and end points). The tolerance specifies how much any one point can be off of the arc that is averaged through all the points.

**Fit Curve Tolerance**



Minimum Number Of Segments

Specifies the minimum number of segments that must be identified before creating an arc. This setting is ignored when the Points option is used.

For more information, see Converting Tessellated Lines to Arcs (page 777)

### Grading Elevation Editor

**NOTE** This property group is displayed when accessing the settings from the GradingElevEditor command.

Use these settings to specify the default behavior for the Elevation Editor, when editing feature lines.

Raise/Lower Increment

Specifies the default vertical distance used for each increment when the feature lines of a grading are raised or lowered to adjust volumes.

Show Grade Breaks Only

Specifies whether only the feature line start/end points and any grade breaks in between are displayed. This option simplifies the editing process by allowing elevation edits to span multiple points.

For more information, see Editing Feature Lines with the Grading Elevation Editor (page 755)

### Grading Layout Tools

**NOTE** This property group is displayed when accessing the settings from the GradingTools command.

Use these settings to specify the default criteria for grading layout.

Grading Criteria Set

Specifies the default Criteria Set to use when creating grading. Click in the Value column, and click  to select a criteria set in the Grading Criteria Set dialog box.

Grading Criteria

Specifies the default Criteria to use when creating grading. Click in the Value column, and click  to select criteria in the Grading Criteria dialog box.

For more information, see Creating Grading (page 790)

## ✅ Grading Volume Tools

**NOTE** This property group is displayed when accessing the settings from the GradingVolumeTools command.

Use these settings to specify the default criteria for grading volumes.

Raise/Lower Elevation Increment

Specifies the default vertical distance used for each increment when the feature lines of a grading are

raised or lowered to adjust volumes. Enter a incremental length in the Value column or click ![icon] and select a length in the drawing area.

Limit Feature Selection To Current Group

Specifies whether features outside the current grading group can be selected for raising or lowering in the volume calculations.

For more information, see Displaying and Adjusting Surface Volumes (page 802)

## ✅ Join Features

**NOTE** This property group is displayed when accessing the settings from the JoinFeatures command.

Use this setting to specify the default tolerance for joining feature lines.

Tolerance

Specifies how close feature lines must be when using the Join (page 771) command.

## ✅ Feature Line Weed

**NOTE** This property group is displayed when accessing the settings from the WeedFeatures command.

Use these settings to specify the default for weeding vertices.

Apply Angle Factor

Specifies whether the Angle factor in the Weed Vertices dialog box is selected by default.

Angle Factor

Specifies the default Angle factor. Enter an angle in the Value column or click ![icon] and select an angle in the drawing area.

Apply Grade Factor

Specifies whether the Grade factor in the Weed Vertices dialog box is selected by default.

Grade Factor

Specifies the default Grade factor. Enter a percentage in the Value column or click ![icon] and select a percentage in the drawing area.

Apply Length Factor

Specifies whether the Length factor in the Weed Vertices dialog box is selected by default.

Length Factor

Specifies the default Length factor. Enter a length in the Value column or click ![icon] and select a length in the drawing area.

Close Point Removal

Specifies whether the 3D Distance factor in the Weed Vertices dialog box is selected by default.

Close Point 3D Distance

Specifies the default 3D Distance factor. Enter a distance in the Value column or click  and select a distance in the drawing area.

For more information, see Weeding Vertices (page 781)

**Related procedures:**

■   Changing Grading Feature Settings (page 736)

# Auto-Balance Volumes Dialog Box

Use this dialog box to specify the required net surface volume (negative for cut or positive for fill) when adjusting the elevation of grading objects.

Required Volume

Specifies the net surface volume for the grading object. When you enter a value and click OK, the grading object is edited so that the required volume is met. The grading object may be adjusted up or down in elevation to meet this value, for example. The action that is performed is recorded in the Description column of the History table in the expanded part of the Grading Volume Tools (page 1899)  dialog box.

# Create Grading Group Dialog Box

Use this dialog box to create a new grading group.

Name

Specifies the name of the new grading group. AutoCAD Civil automatically generates a sequential number.

Description

Specifies an optional description of the grading group.

Automatic Surface Creation

Specifies whether the grading group automatically creates a dynamic surface. If selected, this option activates all of the following fields in this dialog box.

Use the Group Name

Specifies whether to use the grading group name for the name of the dynamic surface. If cleared, the surface is named according to the default name format in the CreateSurface command settings.

Tessellation Spacing

Used for surface creation. Specifies the distance along the footprint to add supplemental breaklines for better definition of the grading group surface.

Tessellation Angle

Used for surface creation. Specifies the angular spacing of supplemental breaklines added around rounded exterior corners, for better definition of the grading group surface.

**Tessellation settings control the placement of breaklines along rounded exterior corners and straight grading segments**



Volume Base Surface

    Specifies the surface to use for comparison with the grading group surface in volume calculations.

**Related procedures:**

■ Using Grading Groups (page 741)

■ Creating a New Grading Group (page 741)

# Grading Criteria Dialog Box

Use this dialog box to manage and create collections of grading criteria for different types of grading.

## Information Tab (Grading Criteria Dialog Box)

Use this tab to change the grading criteria name and description information, and to review details such as when the criteria was most recently modified.

Name

    Specifies the name of the current criteria.

Description

    Specifies a text description of the current criteria.

Created By

    Displays the AutoCAD login name of the person who created the style.

Date Created

    Displays the date and time the style was created.

Last Modified By

    Displays the AutoCAD login name of the person who last modified the style.

Date Modified

    Displays the date and time the style was last modified.

**Related procedures:**

- Using Grading Criteria (page 739)
- Editing Grading Criteria (page 793)

# Criteria Tab (Grading Criteria Dialog Box)

Use this tab to create new grading criteria definitions.

---

**NOTE** The parameters that are displayed under the Grading Method and Projection groups vary, depending on the Target and Projection parameters. For example, if you select Elevation for the target, the Elevation parameter is displayed below Target. You can then enter the elevation.

---

If a criteria is currently used by one or more grading objects, you cannot edit the values for Target or Projection.

## Grading Method

Target

Specifies the target method for the grading. Click one of the following:

- **Surface**: Specifies that the grading projection lines will be extended from the footprint until they match into a surface. You are prompted to select a target surface when you create a grading that uses these criteria.

    **Surface as target (section view)**



- **Elevation**: Specifies that the grading projection lines will be extended from the footprint until they reach a specified elevation. Enter a positive or negative number in the Elevation field.

**Elevation as target (section view)**

fixed elevation as target

footprint

daylight

specified elevation

■ **Relative Elevation**: Specifies that the grading projection lines will be extended from the footprint until they reach an elevation (depth or height) relative to the footprint. Enter a positive or negative number in the Relative Elevation field.

**Relative elevation as target (section view)**

relative elevation as target

footprint

relative elevation

daylight

■ **Distance**: Specifies that the grading projection lines will be extended from the footprint until they are match a specified horizontal distance. Enter a positive number in the Distance field.

**Distance as target (section view)**

horizontal distance as target

footprint

daylight

Projection

Specifies the type of projection. Choices vary according to the target method.

- **Cut/Fill Slope**: Creates the grading by projecting a specific slope value towards a target that can occur both above and below the footprint, such as a surface or elevation target. Slope is always a positive value, as the type of slope determines whether it is going up or down from the footprint. Slope can be formatted as a slope value (run:rise or rise:run), or a grade value (percent or decimal). Related parameter for surface target: see Search Order.

- **Cut Slope**: Creates the grading by projecting a specific slope value up to a target. Slopes can be formatted as a slope value (run:rise or rise:run), or a grade value (percent or decimal).

- **Fill Slope**: Creates the grading by projecting a specific slope value down to a target. Slopes can be formatted as a slope value (run:rise or rise:run), or a grade value (percent or decimal).

- **Distance**: Creates the grading with projection lines that extend a fixed horizontal distance from the footprint.

- **Slope**: Creates the grading by projecting to a specific (absolute) elevation using a specific slope value. Enter a positive or negative value. Slopes can be formatted as a slope value (run:rise or rise:run), or a grade value (percent or decimal).

- **Elevation**: Creates the grading by projecting to a specific (absolute) elevation value.

- **Relative Elevation**: Creates the grading by projecting to an elevation value that is measured relative to the elevation of the footprint.

Search Order

Specifies whether to search first for cut or fill slopes in cases where both would work. For example, where the target surface has a steep slope, both cut and fill slopes may intersect the surface.

- **Cut First**: Creates the grading by trying to grade up from the footprint. If no solution is found, it will grade down.

- **Fill First**: Creates the grading by trying to grade down from the footprint. If no solution is found, it will grade up.

**Different results achieved based on Cut First and Fill First settings (section view)**



Cut First grades up

Fill First grades down

**NOTE** Technically both of these situations are "fill". However, AutoCAD Civil treats the up-slope direction as cut and the down-slope direction as fill.

## (Cut Slope/Fill Slope/Distance/Elevation/Relative Elevation) Projection

**NOTE** The values available for Projection type vary according to the selected Target type.

Format

Specifies how the slope should be represented. Click either Slope or Grade in the list. The format can be further edited in the Grading Settings dialog box (page 1875).

Distance

Specifies the fixed distance from the footprint for the projection. Enter a value, or click  to select a distance in the drawing area.

Slope

Specifies the slope value. Enter a positive value in the form of run:rise or rise:run.

Grade

Specifies the grade value. Enter a positive value as a decimal percent.

## Conflict Resolution

Interior Corner Overlap

Specifies how interior corner projections are cleaned up when the footprint corner has different elevations. This situation results in two possible daylight point elevations. You can choose to average the slopes to reach the same point, or increase or decrease one of the slopes.

**Corner elevation conflict (plan view)**



- **Use Average Slope**: Averages the slopes to reach same daylight point.

- **Hold Slope As Maximum**: Holds the specified grade or slope as the maximum and flattens out the slope projected from the footprint on one side.



- **Hold Slope As Minimum**: Holds the specified grade or slope as the minimum and increases the slope projected from the footprint on one side.

**Related procedures:**

- [Using Grading Criteria](#) (page 739)
- [Editing Grading Criteria](#) (page 793)

# Grading Criteria Set Properties Dialog Box

Use this dialog box to edit name and description properties for the selected criteria set.

Name
> Specifies the name of the current grading criteria set.

Description
> Contains the description of the current grading criteria set.

Created By
> Displays the AutoCAD login name of the person who created the criteria set.

Date Created
> Displays the date and time the criteria set was created.

Last Modified By
> Displays the AutoCAD login name of the person who last modified the criteria set.

Date Modified
> Displays the date and time the criteria set was last modified.

**Related procedures:**

- [Creating Grading Criteria](#) (page 739)

# Grading Editor Dialog Box

Use this dialog box to edit the grading criteria values and change the start or end point of a grading.

You can edit unlocked grading criteria values and apply criteria to grading in the drawing. Editing a value updates the grading in the drawing. Locked values are read-only.

For information about the criteria that you can edit, see the Grading Criteria dialog box (page 1882).

**Select a Grading**

Selects a different grading for editing.

**Related procedures:**

■ Editing Grading Criteria (page 793)

■ Using Grading Criteria (page 739)

# Elevation Editor Dialog Box

Use this dialog box to edit the elevations and grades for feature line, lot line, or survey figure segments. You cannot change the values of the stations or segment lengths.

Green triangles indicate primary geometry points. Green circles indicate intermediate elevation points.

White triangles indicate split points, where two features lines cross with neither having a geometry point at that location. Split points cannot be directly edited. When a vertex shares a point with another intersecting feature line, a small plus symbol is added to the icon for that row. The gray point icons in the grid control and the drawing glyphs represent read-only objects, such as daylight lines and dynamically linked feature lines. For more information, see Editing Feature Lines with the Grading Elevation Editor (page 755).

Use the Shift key to select multiple rows for editing with the Raise/Lower commands.

**Select Line**

Selects a feature line, lot line, or survey figure for editing.

**Zoom To**

Zooms the drawing display to the selected point of intersection (PI) or elevation point.

**Quick Profile**

Creates a quick profile (page 1058) of the feature line.

**Raise/Lower**

Adjusts the elevation of rows either upward or downward. If no rows are selected, prompts for a new elevation for the first point, then adjusts all rows by the same relative amount. If rows are selected, prompts for the new elevation of the first selected row, then adjusts all the selected rows by the same relative elevation.

**Raise Incrementally**

Adjusts the elevations of all points upward by the increment value. If no rows are selected, the option adjusts all points, otherwise it adjusts the points only for the selected rows.

**Lower Incrementally**

Adjusts the elevations of all points downward by the increment value. If no rows are selected, it adjusts all points, otherwise it adjusts the points for the selected rows.

**Set Increment**

Specifies the value to be used by the Raise and Lower commands. Enter the value.

 Flatten Elevations

Specifies that the elevations of all selected rows are flattened to either the same elevation as the first row in the selection, or a constant grade from the start elevation to the end elevation of the selection. Click to open the Flatten dialog box.

 Insert Elevation Point

Inserts an elevation point between the start and end stations of the footprint, creating an intermediate elevation point.

 Delete Elevation Point

Deletes an elevation point between the start and end stations of the footprint. You can delete only a single-row selection of intermediate elevation points.

 Elevations From Surface

Selects an elevation from a surface in the drawing. If there is only one surface, it automatically uses that surface to retrieve the elevations. If there is more then one surface, it prompts you to select the desired surface. If no rows are selected, it updates the elevation of all points. Otherwise, it updates just the selected rows. If the feature line is off the surface, it displays a warning message. If some elevations are updated, but one or more points are off the surface, it displays a message indicating the number of points off the surface that could not be assigned elevations. This option is disabled if the drawing has no surfaces.

 Reverse

Changes the direction of feature lines. Updates the editor so that the order of points is reversed. This command affects the labeling and stationing of feature lines.

 Show Grade Breaks Only

Displays only the feature line start/end points and any grade breaks in between. This option simplifies the editing process by allowing elevation edits to span multiple points. By default, the table displays all feature line points.

 Unselect All Rows

Clears any selected rows so that the Raise, Lower, and Flatten commands to affect the entire length of the footprint.

Station

Identifies the point that starts the current segment.

Elevation

Specifies the elevation of the point identified by the Station.

Length

Displays the length of the current segment.

Grade Ahead

Specifies the end elevation of the current segment in the forward direction. Modifying this grade will change the elevation of the segment. This is the elevation of the next Station point in the next row of the grid.

**NOTE** To edit the grade of multiple rows, select the desired rows and edit the grade of one of the selected rows. The new grade will be applied to all selected rows.

Grade Back

Specifies the grade of the current segment from its end to start. Modifying this grade will change the start elevation of the segment, which is the point at the beginning of the current row.

**Related procedures:**

- [Editing the Elevations of a Feature Line](#) (page 753)

# Elevation Editor - Flatten Dialog Box

Use this dialog box to edit the elevations and grades for feature line, lot line, or survey figure segments to either a constant elevation or a constant grade.

Select **Constant Elevation** to modify the points to a constant elevation. If no rows are selected it flattens the entire feature line using the elevation of the start point. If some rows are selected it flattens just those using the elevation of the first selected row. Select **Constant Grade** to edit the selected rows to a constant grade. The entire feature line grade is modified if no rows are selected. It holds the start elevation and the end elevation and sets the points in between to the same grade, in effect eliminating the grade breaks.

Select Constant Elevation to select multiple rows for editing with the Raise/Lower commands.

**Related procedures:**

- [Editing the Elevations of a Feature Line](#) (page 753)

# Grading Group Properties Dialog Box

Use this dialog box to view and edit the properties of the selected grading group.

**See also:**

- [Grading Properties](#) (page 799)

# Information Tab (Grading Group Properties Dialog Box)

Use this tab to view or change general information for the grading group. The options for automatic surface creation and volume base surface are usually set when the grading group is created, but they can be changed here.

Name

Specifies the name of the grading group.

Description

Specifies an optional description of the group.

Automatic Surface Creation

Specifies whether the grading group automatically creates and updates a dynamic surface. Default surface name is the grading group name.

- **Selected**: If the check box is already selected, it means a surface exists for the grading group. You can clear the check box to either delete the dynamic surface or to create a detached surface. When you clear the check box, you are prompted to delete the surface. If you click No, the surface is detached from the grading group.

- **Cleared**: If the check box is already cleared, no dynamic surface exists for the grading group. You can select the check box to create a dynamic surface, and the Create Surface dialog box is opened.

Tessellation Spacing

Specifies the gap between projection lines on the grading group surface.

Tessellation Angle

Specifies the angular distance between radial projection lines on exterior corners of the grading group surface.

Volume Base Surface

Specifies the surface to use for comparison with the grading group surface in volume calculations.

**Related procedures:**

■ Create Grading Group Dialog Box (page 1881)

■ Using Grading Groups (page 741)

## Properties Tab (Grading Group Properties Dialog Box)

Use this read-only tab to review properties of the grading group.

You can see basic statistics such as

■ Area covered by the grading group

■ Range of elevations and slopes

■ Volume report

■ Number of times the grading group used specific grading criteria and styles

**Related procedures:**

■ Grading Group Properties (page 799)

## Grading Creation Tools

Use this toolbar to quickly create new grading, apply grading criteria and styles, and edit existing grading.

Set The Grading Group

Opens the Select Grading Group dialog box (page 1900). Select the current grading group, or create a new group.

Set The Target Surface

Opens the Select Surface dialog box. Select the surface to use as a target. Applies only when the Grading Criteria uses Surface as the Target.

Set The Grading Layer

Opens the Set Grading Layers dialog box. Specify on which layer the grading should be created.

Select a Criteria Set

Sets the current criteria set, from which you can select specific criteria.

Select a Criteria

Click the Down arrow to select a criteria to apply to the grading.

Style Picker

Use these options to edit the current criteria or create a new criteria.

## Grading Creation Tools

Create Grading

Creates grading by using prompts on the command line. The grading uses the current style and criteria.

Copy Create Grading

Creates a grading by copying the criteria and style from an existing grading object.

Create Transition

Creates a transitional slope between two different criteria or two different values.

Create Infill

Creates a grading face that has no criteria to fill in any holes that are left by creating grading.

## Grading Editing Tools

Edit Grading

Provides a series of command line prompts for you to change the criteria of a grading object.

Delete Grading

Deletes a grading and removes it from the grading group.

Change Grading Group

Prompts you to select the grading objects and then opens the Select Grading Group dialog box (page 1900). Select a grading group to move the grading objects into.

## Grading Utilities

Grading Volume Tools

Opens the Grading Volume Tools (page 1899). Raise or lower the grading group to meet target cut and fill volumes.

Create Detached Surface

Creates a new surface that is not associated with the grading group, and therefore does not update to reflect changes in the group.

Grading Editor

Prompts you to select an existing grading and opens the Grading Editor dialog box.

Grading Elevation Editor

Prompts you to select an existing feature line or lot line, then displays the Grading Elevation Editor dialog box.

Grading Group Properties

Opens the Grading Group Properties dialog box (page 1891).

Grading Properties

Opens the Grading Properties dialog box (page 1894). Note that this is the only place to view the properties of an individual grading.

Expand

Shows or hides the current grading criteria values and the style selectors. You can view the criteria values here.

**Related procedures:**

- Creating Grading (page 790)

# Grading Properties Dialog Box

Use this dialog box to view and change the styles and properties for the selected grading.

Grading Group

Specifies the grading group for the selected grading.

Style Name

Specifies the grading style for the selected grading.

Property & Value

Displays information about the grading type and grading criteria.

**Related procedures:**

- Grading Object Properties (page 800)

# Grading Style Dialog Box

Use this dialog box to control the display of each grading component.

# Information Tab (Grading Style Dialog Box)

Use this tab to change the grading style name and description information, and to review other details, such as when the style was most recently modified.

Name

Specifies the name of the current grading style.

Description

Displays a description of the current grading style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

**Related procedures:**

■ Using Grading Styles (page 737)

## Center Marker Tab (Grading Style Dialog Box)

Use this tab to specify the size of the triangular marker for the center of a grading.

Size

Specifies the size of the marker and how it is calculated:

■ **Percentage Of Screen**: Marker size is a fixed percentage of the size of the drawing window, and resizes for the current zoom level. Default option.

■ **Plotted Size**: Marker size is based on the current drawing scale (inches or millimeters).

■ **Fixed Size**: Marker size is specified in the current drawing units (usually feet or meters).

**Related procedures:**

■ Using Grading Styles (page 737)

## Slope Pattern Tab (Grading Style Dialog Box)

Use this tab to configure the pattern used to mark grading slopes.

Slope patterns are not applied to infill gradings.

Slope Pattern

Specifies whether to use a slope pattern with the current grading style. Select a slope pattern style and edit a style or create a new one.

Slope Range

Specifies whether the slope pattern is applied to a limited range of slope values, and specifies the range.

**Related procedures:**

■ Using Grading Styles (page 737)

## Display Tab (Grading Style Dialog Box)

Use this tab to specify the defaults for the separate components of a grading object.

View Direction

Specifies the type of view: either 2D or 3D.

Component

Specifies the name of the grading component.

Visible

Controls component visibility.

Layer
    Opens the AutoCAD Layer Selection dialog box. You can assign components to a layer that you create during drawing setup or you can click the New button and create a new layer for the component.

Color
    Specifies the component's color. You can select a color for the component from the Select Color dialog box, or you can specify that the component should derive its color BYLAYER or BYBLOCK.

Linetype
    Displays the Select Linetype dialog box, which allows you to specify the linetype for the component.

Lineweight
    Displays the Select Lineweight dialog box, which allows you to specify the lineweight for the component.

Linetype Scale
    Specifies the linetype scale for the component.

Plot Style
    Sets the plot style for the component.

**Related procedures:**

■ Using Grading Styles (page 737)


## Summary Tab (Grading Style Dialog Box)

Use this tab to view summary information about the grading. You can edit the Name and Description.

**Information**

Name
    Specifies the name of the current grading style.

Description
    Specifies the description of the current grading style.

Created By
    Displays the AutoCAD login name of the person who created the style.

Date Created
    Displays the date and time the style was created.

Last Modified By
    Displays the AutoCAD login name of the person who last modified the style.

Date Modified
    Displays the date and time the style was last modified.

**Related procedures:**

■ Using Grading Styles (page 737)


# Feature Line Style Dialog Box

Use this dialog box to control the display of each grading feature line component.

# Information Tab (Feature Line Style Dialog Box)

Use this tab to change the grading style name and description information, and to review other details, such as when the style was most recently modified.

Name

Specifies the name of the current feature line style.

Description

Displays a description of the current feature line style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

**Related procedures:**

■ Applying Feature Line Styles (page 746)

# Profile Tab (Feature Line Style Dialog Box)

Use this tab to specify the appearance of marker symbols for a feature line projected into a profile view.

You can specify different styles for the markers at the beginning, internal vertices, and the end of the feature line. Click  to edit the style. Click  to preview the selected style.

**Related procedures:**

■ Applying Feature Line Styles (page 746)

# Section Tab (Feature Line Style Dialog Box)

Use this tab to specify the appearance of the feature line marker symbol in a section view.

**Related procedures:**

■ Applying Feature Line Styles (page 746)

# Markers Tab (Feature Line Style Dialog Box)

Use this tab to specify the appearance of the feature line marker symbol in the drawing. This marker symbol is for displaying the feature line in a section view.

Marker Style

Select the check box and specify a marker style in the list. Click  to edit the style. Click  to preview the selected style.

**Related procedures:**

■ Applying Feature Line Styles (page 746)

## Display Tab (Feature Line Style Dialog Box)

Use this tab to specify the default display for the components of a feature line.

View Direction

Specifies the context for which you are setting the display:

■ Plan – the printed drawing

■ Model – the on-screen design window

■ Profile – projected into a profile view (line and vertex markers)

■ Section – projected into a section view (crossing markers)

Component Type

Specifies the name of the feature line component.

Visible

Controls component visibility.

Layer

Opens the AutoCAD Layer Selection dialog box. You can assign components to a layer that you create during drawing setup or you can click the New button and create a new layer for the component.

Color

Specifies the component's color. You can select a color for the component from the Select Color dialog box, or you can specify that the component should derive its color BYLAYER or BYBLOCK.

Linetype

Displays the Select Linetype dialog box, which allows you to specify the linetype for the component.

LT Scale

Specifies the linetype scale for the component.

Lineweight

Displays the Select Lineweight dialog box, which allows you to specify the lineweight for the component.

Plot Style

Sets the plot style for the component.

**Related procedures:**

■ Using Grading Styles (page 737)

## Summary Tab (Feature Line Style Dialog Box)

Use this tab to view summary information about the feature line. You can edit the Name and Description.

**Information**

Name

Specifies the name of the current feature line style.

Description

Specifies the description of the current feature line style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

**Related procedures:**

■  Using Grading Styles (page 737)


# Grading Volume Tools

Use this toolbar to adjust the cut and fill volumes for a grading group.

Controls on this toolbar are not available (dimmed) if any required data is unavailable. For example, this can occur if a grading group has not been selected, or if one of the required surfaces has not been defined.

Set the Grading Group

Click to specify the grading group to adjust. The name of the group is displayed along the bottom of the toolbar.

Grading Group Properties

Opens the Grading Group Properties dialog box (page 1891).

Entire Group

Click to adjust the elevation of the whole grading group.

Selection

Click to select one or more features. Click to select the features to adjust.

Raise the Grading Group

Raises the elevation of the grading group by the amount in the box.

Lower the Grading Group

Lowers the elevation of the grading group by the amount in the box.

Automatically Raise/lower to Balance Volumes

Opens the Auto-Balance Volumes dialog box (page 1881). Specify a target value for net volume and automatically balance cut and fill volumes to approach the target.
Auto-balancing volumes to the target is an iterative process due to surface irregularities. The system aims to reach the target volume, with a tolerance of 0.1% of the difference between the starting and target volumes. If necessary, re-run the command to get closer to the target.

**NOTE** This button is unavailable unless the Automatic Surface Creation setting is selected and a Volume Base Surface is specified in the Grading Group Properties  to display the Grading Group Properties dialog box.

 **Expand the Grading Volume Tool**

Shows or hides the history of cut and fill adjustments. This history is erased when you close the toolbar.

Volume Display

Displays the current cut and fill volumes and the resulting net requirements for the grading group. This display updates whenever you modify either of the two surfaces involved in the comparison.

**Related procedures:**

■ Editing the Elevations of a Feature Line (page 753)

# Insert PVI Dialog Box

Use this dialog box to display and edit the values of individual PVIs that you add to a grading feature line. This dialog box is displayed when you click the Insert PVI button in the Elevation Editor dialog box and select a feature line.

Station

Specifies the station of the new PVI. Edit the station definition if needed.

Elevation

Specifies the elevation of the PVI. Edit the elevation if needed.

**Related procedures:**

■ Editing PVIs (page 1093)

# New Slope Pattern Component Dialog Box

Use this dialog box to specify the position of a new component in relation to existing slope pattern components.

Components are numbered in left to right order.

Insert New Component At Position

Specifies the position at which the new component is placed. A new component is placed at the end of the series by default. If you select a different position, existing components in that and higher positions are incremented by one.

**Related procedures:**

■ Creating Grading Styles (page 737)

# Select Grading Group Dialog Box

Use this dialog box to select the grading group to which an individual grading belongs.

Site Name

Select the site you want from the list of the sites in the current drawing. Click  to select a site in the drawing.

Group Name

Select the group you want from the list of the grading groups in the current drawing. Click ![icon] to select a grading group in the drawing. Click ![icon] to create a new grading group.

Description

Displays the description of the grading group, if one has been defined.

**Related procedures:**

- Using Grading Groups (page 741)
- Changing the Grading Group of a Grading Object (page 742)

# Set Grading Layers Dialog Box

Use this dialog box to override the layer settings for the grading objects you create.

Grading Layer

Specifies the layer name the grading is on.

The Object Layer dialog box is used to specify default base layers for drawing objects. The base layer is where the object physically resides.

![icon]

Opens the Object Layers dialog box, which allows you to specify the layer.

# Slope Pattern Style Dialog Box

Use this dialog box to specify the format of a slope pattern.

# Information Tab (Slope Pattern Style Dialog Box)

Use this tab to change the style name and description information, and to review other details, such as when the style was most recently modified.

Name

Specifies the name of the current slope pattern style.

Description

Specifies the description of the style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

**Related procedures:**

■ Using Grading Styles (page 737)

# Layout Tab (Slope Pattern Style Dialog Box)

Use this tab to configure the appearance of a slope pattern.

Each line in the slope pattern is a separate component.

Component

Specifies the name of the slope pattern component represented in the table of parameter values. You can use the buttons next to this field to add, copy, or delete a component.

 Add Component

Creates a new component using the default parameters. Opens the New Slope Pattern Component dialog box. Specify the position of the new component in relation to existing components.

 Copy Component

Creates a new component using the same parameters as the current component. Opens the New Slope Pattern Component dialog box (page 1900). Specify the position of the new component in relation to existing components.

 Delete Component

Removes the current component from the slope pattern. Not available if only one component exists.

## Slope Line Parameters

Pattern Start

Specifies whether the pattern begins at the top or bottom of the slope.

Length Type

Specifies how to determine the line length for this component. Each option invokes one or more additional parameters, as described:

■ **Length**: Specifies that Line length is a fixed value. Related parameter Length specifies the value in drawing units (usually feet or meters). The line is trimmed if the slope is shorter than this length.

■ **Percent of Length**: Specifies that Line length is a percentage of the slope length, up to a specified maximum. Related parameters: Percent of Length specifies the percentage value; Maximum Length specifies the maximum line length in drawing units.

■ **Slope Ratio**: Specifies that line length varies directly with the steepness of the slope. Related parameters: Slope 1 and Slope 2 specify outside limits for slope values that are represented by line length; Percent of Length 1 specifies the line length as a percent of slope length for Slope 1; Percent of Length 2 specifies the same value for Slope 2. Example: Slope 1 = 6:1, Percent of Length 1 = 20%, Slope 2 = 2:1, Percent of Length 2 = 80%. On a 10-feet long slope, a 6:1 slope is marked with a 2-feet line and a 2:1 slope is marked with an 8-feet line. An intermediate value, 4:1, is marked with a 5-feet line. An outside value, 10:1, is marked with a 1-foot line.

Color

Specifies the color of the line.

Linetype

Opens the Select Linetype dialog box. Specify the linetype for the component.

Lineweight

    Opens the Select Lineweight dialog box. Specify the lineweight for the component.

## Slope Line Offset Parameters

Offset Type

    Specifies how to determine the offset distance between this component and the previous component in the slope line style. Distances are measured as a linear value along the feature and an angular value around exterior corners. Each option invokes additional parameters as described.

■ **Percent of Length**: Specifies that Offset is a percentage of the slope length at the position being offset from. Related parameters: Percent of Length specifies the percentage; Minimum Distance and Maximum Distance specify values for minimum and maximum offsets in drawing units; Radial Offset Angle specifies the angular value.

■ **Distance**: Specifies that Offset is a fixed value. Related parameters: Distance specifies the linear value in drawing units (usually feet or meters); Radial Offset Angle specifies the angular value. The Radial Offset Angle parameter is not used in corridors.

■ **Divide**: Specifies that Offset evenly divides the distance between the previous and the next components to insert one or more instances of the current component. This option is not available if the previous or next component is set to Divide. Related parameter: Number of Lines specifies the number of instances of this component to insert.

## Slope Line Symbol

Symbol Type

    Specifies whether a symbol is added to the beginning of the slope line, and if so, the type of symbol:

■ **None**: No symbol.

■ **AutoCAD Block**: Adds a block that is defined in the drawing. Related parameter: Block Name specifies the block to insert.

■ **Triangle**: Adds a triangle.

■ **Filled Triangle**: Adds a triangle with a solid fill.

■ **Tapered Lines**: Adds a set of lines parallel to the edge of the slope, tapering toward the slope line in a triangular pattern. A gap exists between the last tapered line and the slope line. Related parameter: Number of Lines specifies the number of lines to include in the set.

■ **Tapered Lines (No Gap)**: Adds a set of tapered lines, and attaches the last tapered line to the slope line. Related parameter: Number of Lines specifies the number of lines to include in the set.

Length Type

    Specifies how to determine the length of the symbol. Each option invokes an additional parameter:

■ **Length**: Specifies that Symbol length is a fixed value. Related parameter Length specifies the value in drawing units (usually feet or meters).

■ **Percent of Length**: Specifies that symbol length is a percentage of the slope length. Related parameter Percent of Length specifies the percentage value.

Width Ratio

    Specifies the ratio of the symbol's width to its length along the slope line (width = length * width ratio).

Color

    Specifies the color of the symbol.

Linetype

Opens the Select Linetype dialog box. Specify the linetype for the symbol.

Lineweight

Opens the Select Lineweight dialog box. Specify the lineweight for the symbol.

**General Properties**

Minimum Display Length

Specifies the minimum slope length on which to display the slope pattern.

Use the following parameters to configure the sample slope in the preview window.

Preview Feature Length

Specifies the length of feature line to display.

Preview Slope Length

Specifies the length of slope to display.

Preview Slope

Specifies the slope value to display.

**Related procedures:**

■ Using Grading Styles (page 737)


# Edit Geometry Panel

Use this panel to edit feature lines, 2D and 3D polylines, survey figures, and parcel lines.

This is a standard panel that you access by selecting Modify tab ➤ Edit Geometry panel.

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**NOTE** See Editing Feature Lines (page 751) for a list of supported objects for each command. Supported objects are also listed at the command line when you run the command.

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Break

Breaks a feature line into two feature lines. (page 784)

Trim

Trims a feature line. (page 785)

Join

Joins multiple feature lines, creating a single object. (page 771)

Reverse

Reverses the direction of feature lines for stationing purposes. (page 772)

Fillet

Rounds the corners of feature lines. (page 776)

Fit Curve

Converts tessellated curves to true arcs for better grading results. (page 777)

 Smooth

Smooths out jagged feature lines. (page 779)

 Weed

Deletes vertices and elevation points from feature lines. (page 781)

 Stepped Offset

Offsets feature lines at an elevational difference. (page 783)

# Edit Elevations Panel

Use this panel to edit feature lines, 2D and 3D polylines, survey figures, and parcel lines.

This is a standard panel that you access by selecting Modify tab ➤ Edit Elevations panel.

**NOTE** See Editing Feature Lines (page 751) for a list of supported objects for each command. Supported objects are also listed at the command line when you run the command.

 Elevation Editor

Edits feature line elevations using the Elevation Editor. (page 753)

 Quick Elevation Edit

Edits the elevations of a feature line by snapping to points that you can edit in the drawing. (page 754)

 Edit Elevations

Edits feature-line or lot-line elevations, using the command line. (page 753)

 Set Grade/Slope Between Points

Specifies the grade or slope between selected points on a feature line. (page 758)

 Set Elevation By Reference

Sets the feature line elevation in relation to an existing elevation in the drawing. (page 759)

 Insert Elevation Point

Inserts an elevation point (page 765)

 Delete Elevation Point

Deletes elevation points. (page 767)

 Insert High/Low Elevation Point

Inserts high or low elevation points into feature lines. (page 767)

 Raise/Lower

Raises and lowers feature line elevations. (page 764)

**Elevations From Surface**

Creates feature-line elevations from a surface. (page 768)

**Insert PI**

Inserts points of intersection. (page 769)

**Delete PI**

Deletes points of intersection. (page 771)

**Set Elevation By Reference**

Sets an elevation by referencing from a selected point. (page 759)

**Raise/Lower Elevation By Reference**

Raises or lower elevations by referencing a selected point. (page 761)

**Adjacent Elevations By Reference**

Sets elevations by referencing an adjacent feature line. (page 762)

**Break**

Breaks a feature line into two feature lines. (page 784)

**Trim**

Trims a feature line. (page 785)

**Join**

Joins multiple feature lines, creating a single object. (page 771)

**Reverse**

Reverses the direction of feature lines for stationing purposes. (page 772)

**Fillet**

Rounds the corners of feature lines. (page 776)

**Fit Curve**

Converts tessellated curves to true arcs for better grading results. (page 777)

**Smooth**

Smooths out jagged feature lines. (page 779)

**Weed**

Deletes vertices and elevation points from feature lines. (page 781)

**Stepped Offset**

Offsets feature lines at an elevational difference. (page 783)

# Feature Line Contextual Commands

Use these commands to create and edit feature line properties.

You can access these commands by selecting a feature line and then select Feature line tab ➤ Modify panel.

**NOTE** See Editing Feature Lines (page 751) for a list of supported objects for each command. Supported objects are also listed at the command line when you run the command.

Feature Line Properties
   Draws a feature line. (page 748)

Apply Feature Line Names
   Applies optional names to feature lines. (page 746)

Apply Feature Lines Styles
   Applies styles to feature lines. (page 746)

Remove Dynamic Links
   Removes feature line dynamic links to alignments and corridors. (page 747)

# Create Feature Lines Dialog Box

Use this dialog box to specify feature line settings when drawing a feature line or creating a feature line from objects.

Site

   Specifies the site in which to place the feature line. Select a site in the list or click  to select a site from the drawing.

Name

   Specifies the name of the feature line. To name the feature line, click on its default name and enter a new name or use the naming template. Click  to open the Name Template Dialog Box (page 1806), where you can modify the feature line naming template.
   Feature line names are optional and can be used to name significant feature lines in the drawing. When there are named feature lines you can select feature lines to edit by name from a list.

Style

   When selected, specifies a feature line style. Select a style in the list. Click  to edit the style. Click  to preview the selected style. A style is optional for a feature line, but it provides useful display options and allows you to control color, visibility, and other settings. A style is also used to set the control the feature line split point resolution.

Layer
   Opens the Object Layer dialog box, where you can specify the layer the feature line should be created on. This layer cannot be specified if Use Selected Entity Layers is selected under Conversion Options.

Use Current Layer
   Places the new feature lines on the current layer.

Use Selected Entity Layer

Specifies the layer where the converted feature lines are placed.

- ■ **Selected**: Places feature lines on the same layer as the selected objects.

  NOTE This can make it difficult to select the feature line if the Erase Existing Entities check box is cleared.

- ■ **Cleared**: Places the feature lines on the layer specified by the Layer setting above.

## Conversion Options

Erase Existing Entities

Specifies whether the selected objects are erased when they are converted to feature lines.

- ■ **Selected**: Erases the selected objects from the drawing.
- ■ **Cleared**: Does not erase the selected objects.

Assign Elevations

Specifies whether to assign elevations to the entities being converted.

- ■ **Selected**: Click OK to open the Assign Elevations dialog box.
- ■ **Cleared**: The Assign Elevations dialog box is not opened.

Weed Points

Specifies whether the selected objects are weeded when they are converted to feature lines.

- ■ **Selected**: Click OK to open the Weed Vertices dialog box.
- ■ **Cleared**: The Weed Vertices dialog box is not opened.

**Related procedures:**

- ■ Creating Feature Lines (page 743)
- ■ Weeding Vertices (page 781)

# Assign Elevations Dialog Box

Use this dialog box to assign elevations when creating feature lines from objects. You can access this dialog from the Create Feature Lines dialog box.

These options override the elevations of the objects that you are converting to feature lines.

Elevation

Assigns a fixed elevation to the feature line.

From Gradings

Creates a temporary surface from selected gradings to compute the feature line elevation. This produces the same result as selecting the grading group surface if you have not created a grading group surface.

From Surface

Assigns elevations to the feature line from a surface in the drawing.

Select Intermediate Grade Break Points

Inserts intermediate grade breaks when the entity crosses surface TIN lines, creating elevation points at these locations.

**Related procedures:**

■

# Create Feature Line From Alignment Dialog Box

Use this dialog box to specify feature line settings when creating a feature line from an alignment.

Alignment

Displays the name of the source alignment.

Site

Specifies the site in which to place the feature line. Select a site in the list or click  to select a site in the drawing.

NOTE The feature line can not be created in the same site as the alignment. If the alignment is in a site, that site will not be available in the site list.

Name

Specifies the name of the feature line. Select the check box and click  to open the Name Template dialog box. In addition to the style and counter properties, there are properties to include the alignment and profile names in the feature line name.

Profile

Specifies a vertical alignment from which to acquire the elevations for the feature line. Click  to select a profile in the drawing. If the alignment does not have a vertical alignment, you can create the feature line only while the dynamic link option is disabled. For the dynamic link, a vertical alignment is required.

Mid-ordinate Distance

Sets the vertical curve tessellation factor.

Spiral Tessellation Factor

Sets the clothoid spiral tessellation factor.

Weed Points

Specifies whether to open the Weed Vertices dialog box so you can weed the objects when converting to a feature line. Change this value to True to open the Weed Vertices dialog box.

Style

Select the check box and specify a feature line style in the list. Click  to edit the style. Click  to preview the selected style. A style is optional for a feature line, but it provides useful display options, allowing you to control color, visibility, and other settings.

 Layer

Opens the Object Layer dialog box. Specify the layer on which the feature line should be created. This layer cannot be specified if Use Selected Entity Layers is selected under Conversion Options.

Create Dynamic Link to the Alignment

Creates a dynamic link to the alignment so that the feature line updates dynamically with any changes to the alignment or the selected vertical alignment. If the feature line is created with the dynamic link,

the link can be disabled, using the Feature Line Properties (page 1910) dialog box. If the dynamic link is disabled, it cannot be restored.

■ **Selected**: The feature line is linked to the alignment.

■ **Cleared**: The feature line is not linked to the alignment and can be edited with any of the feature line editing commands.

**Related procedures:**

■ Creating Feature Lines (page 743)

■ Creating Feature Lines from Alignments (page 745)

# Feature Line Properties Dialog Box

## Information Tab (Feature Line Properties Dialog Box)

Use this dialog box to assign a style to a feature line or to change the style assigned to a feature line.

Name

■ Specifies that a name is applied to the feature line. Click  to edit the name template.

Style check box

■ **Selected**: Specifies that a style is applied to the feature line.

■ **Cleared**: Style selection options are not available.

Style

Specifies the style for the feature line.



Specifies the style options. You can create a new style, copy or edit the current style selection, or pick a style from drawing.



Opens the Style Detail dialog box where you can preview the style and creation information.

Dynamic Link

■ Displays the name of the alignment and profile to which the feature line is linked. You can change the profile that is used to set the elevations and the mid-ordinate distance can be changed.

Mid-ordinate Distance

Specifies the mid-ordinate distance for tessellating curves into short line segments. The default is 0.100.

Weed Points

Specifies whether to open the Weed Vertices dialog box so you can weed the objects when converting to a feature line. Change this value to True to open the Weed Vertices dialog box.

**Related procedures:**

■ Feature Line Properties (page 801)

## Statistics Tab (Feature Line Properties Dialog Box)

This dialog box displays read-only feature line statistics.

Count
  Displays the total number of points.

PI Points
  Displays the total number of PI points.

Elevation Points
  Displays the total number of intermediate elevation points.

2D Length
  Displays the total 2D length.

3D Length
  Displays the total 3D length.

Minimum Elevation
  Displays the minimum elevation.

Maximum Elevation
  Displays the maximum elevation.

Minimum Grade
  Displays the minimum grade.

Maximum Grade
  Displays the maximum grade.

### Breakline Data

If the feature line is used by any surfaces as breakline data, the following list is displayed.

Surface
  Displays the name of the surface.

Breakline Group
  Displays the breakline group.

### Related procedures:

■ Adding a Feature Line to a Surface as a Breakline (page 786)

# Weed Vertices Dialog Box

Use this dialog box to specify settings for removing vertices and elevation points from feature lines, parcel lines, and 2D or 3D polylines.

### Weeding Factors

Select one or more weeding factor to apply. Larger values remove more points. If more than one factor is specified, both factors have to be met before the point is selected for weeding.

When you change a value, the display in the drawing is updated with red glyphs to show the points that will be weeded.

Angle
  Specifies the weeding angle.

weeding angle factor (plan view)

possible vertex to be
deleted

if angle < angle factor, then the vertex will be deleted

**Grade**

Specifies the weeding grade.

weeding grade factor (profile view)

G1 %

G2 %

possible vertex to be deleted

if the difference between G1 and G2 < grade factor,
then the vertex will be deleted

**Length**

Specifies the weeding distance. Enter the distance value or click to select it in the drawing.

weeding length factor (plan view)

possible vertex to be deleted

L

if L < length factor, then the vertex will be deleted

## Close Point Removal

**3D Distance**

Specifies the 3D distance for weeding. Points that are close together in plan view, with slightly different elevations can be weeded using this setting.

weeding 3D distance factor (profile view)



D

possible vertex to be deleted

if D is < 3D distance factor, then the vertex will be deleted

**Related procedures:**

■ Weeding Vertices (page 781)

# Fit Curve Dialog Box

Use this dialog box to specify the tolerance and minimum number of segments for applying the Fit Curve command.

Tolerance

Specifies the maximum distance from a PI along the feature to the arc that is to be inserted. When an arc is inserted, it should pass through all the feature line points (between the start and end points). The tolerance specifies how much any one point can be off of the arc that is averaged through all the points.

**Fit Curve Tolerance**



Minimum Number Of Segments

Specifies the minimum number of segments that must be identified before creating an arc. This setting is ignored when the Points option is used.

**Related procedures:**

■ Converting Tessellated Lines to Arcs (page 777)

# Set Elevations From Surface Dialog Box

Use this dialog box to select the surface from which to obtain feature line elevations.

Surface Selection List

Select a surface in the list or click  to select it in the drawing.

Insert Intermediate Grade Break Points

Inserts an elevation point (page 2473) at each location where the feature line crosses a surface TIN line.

**Related procedures:**

■   Setting Feature Line Elevations from a Surface (page 768)

# Statistics Tab (Feature Line Site Properties Dialog Box)

Use this tab to view the combined statistics for all feature lines in the site. This data cannot be edited.

Feature Lines

Displays the total number of feature lines.

PI Points

Displays the total point of intersection (PI) points.

Elevation Points

Displays the total elevation points.

2D Length

Displays the total 2D length.

3D Length

Displays the total 3D length.

Minimum Elevation

Displays the minimum elevation.

Maximum Elevation

Displays the maximum elevation.

Minimum Grade

Displays the total point of intersection (PI) points.

Maximum Grade

Displays the maximum grade.

# Grouped Statistics Tab (Feature Line Site Properties Dialog Box)

Use this tab to view the combined statistics for all feature lines in the site. This data cannot be edited.

The data is grouped in two list boxes allowing you to extract data based on layer and/or style. The following information is listed for each style or layer.

Count

Displays the total number of feature lines.

2D Length

Displays the total 2D length.

**3D Length**

Displays the total 3D length.

**Minimum Elevation**

Displays the minimum elevation.

**Maximum Elevation**

Displays the maximum elevation.

**Minimum Grade**

Displays the total point of intersection (PI) points.

**Maximum Grade**

Displays the maximum grade.

## Options Tab (Feature Line Site Properties Dialog Box)

Use this dialog box to determine feature line style priority.

When two feature lines with different styles intersect, the one with the higher set priority determines the elevation. Feature lines without a style have the lowest priority. If two feature lines have the same style or no style, then priority is given to the last edited feature line.

**Feature Line Crossings: Split Point Resolution**

Displays the feature line style priority from highest to lowest. You can drag and drop styles in the list or use the up/down arrows to specify priority. When new styles are created, they are added to the end of the list.

**Filter Unused Styles**

Filters out unused styles and displays styles referenced in the current site. You can edit the style priority in the filtered state, but some styles may be hidden. You can move a style between two visible styles that have been filtered. If the style moves to a lower priority, it is placed above the filtered styles immediately following the style it was placed after. If the style was moved up, it is placed below the filtered styles, immediately above the style it was moved ahead of.

**Copy From Site**

Selects a different site in the drawing in order to copy the style priority order. If there is only one site in the drawing, this option is unavailable.

**Apply to All Sites**

Applies the current style priority order to all sites in the drawing. If there is only one site in the drawing, this option is unavailable.

**Save as Default**

Sets the default command setting to the currently set priority.

## Apply Feature Line Names Dialog Box

Use this dialog box to edit the names for a selection set of feature lines.

**Name**

Toggles the feature line names on or off. If only a single feature line was edited, you can use the name

template, or type in a new name. Click  to access the Name Template Dialog Box (page 1806), where you can modify the feature line naming template. If multiple feature lines are selected, use the name template to apply unique names to the selection set. If you attempt to use a name template that does not include the counter, an error message will display.

**Related procedures:**

■ Applying Feature Line Names (page 746)

# Apply Feature Line Styles Dialog Box

Use this dialog box to specify a style for a selection set of feature lines.

Style List

Specifies the following options for style selection:

■ Create New

■ Copy Current Selection

■ Create Child of Current Selection (Labels only)

■ Edit Current Selection

■ Pick from Drawing

Specifies the style options. You can create a new style, copy or edit the current style selection, or pick a style in the drawing.

Opens the Style Detail dialog box, where you can preview the style and creation information. You can also use the checkbox to turn off styles for the selected feature lines.

**Related procedures:**

■ Applying Feature Line Styles (page 746)

# Edit Feature Line Curve Dialog Box

Use this dialog box to edit the radius of a feature line (or survey figure) arc. You can also delete the arc from the feature line.

These options override the elevations of the objects you are converting to feature lines.

Radius

Specifies the radius of the arc.

Length

Displays the length of the arc.

Chord

Displays the length of the chord.

Tangent

Displays the length of the tangent.

Maintain Tangency

When the curve is not tangent, the Maintain Tangency button is activated. If you toggle this button the curve will be adjusted, to making it tangent to the adjacent segments. If the curve is tangent, the button is disabled.

**Edit Previous Curve**

Moves to the previous curve on the feature line. If the current curve has been edited it will be saved before moving to the previous curve. If there is no previous curve, this option is disabled

**Edit Next Curve**

Moves to the next curve on the feature line. If the current curve has been edited, it will be saved before moving to the next curve. If there is no following curve this option is disabled.

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**NOTE** The left and right arrow keys can also be used to navigate the curves. You can edit the radius, then press the arrow key to move without using the mouse.

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**Pick Curve Radius**

Selects points in the drawing to specify the curve radius. If the radius is valid, the value appears in the Radius edit box and the arc preview updates. If the radius is too large for the feature line, an error message is displayed.

# Intersections Dialog Boxes

# 54

## Edit Feature Settings - Intersections Dialog Box

Use this dialog box to view and change intersection-specific settings.

This topic documents settings in all intersection-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the ▯ drawing icon.

- Intersection feature settings are listed after the General and Labeling property groups, and are identified by the ✚ intersection icon.

- Intersection command settings are identified by the 🗹 command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### 🗹 Assembly Insertion Defaults

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the defaults assigned during intersection object creation. For more information, see Changing Corridor Assembly Frequency (page 1451).

Frequency Along Tangents
  Specifies the frequency to insert assemblies along tangent segments of the intersection area corridor baseline.

Frequency Along Curves
  Specifies the frequency to insert assemblies along curved segments of the intersection area corridor baseline.

Frequency Along Spirals
  Specifies the frequency to insert assemblies along spiral segments of the intersection area corridor baseline.

Horizontal Geometry Points

Specifies whether to add assemblies at horizontal geometry points of the corridor baselines in the intersection area.

Superelevation Critical Points

Specifies whether to add assembies atsuperelevation critical points of the corridor baselines in the intersection area.

Profile Geometry Points

Specifies whether to add assembies at profile geometry points of the corridor baselines in the intersection area.

Profile High Low Points

Specifies whether to add assemblies to the highest and lowest points of the profile to define intersection corridor.

Frequency Along Profile Curves

Specifies the frequency that intersection object assemblies are inserted along curved segments of the profile.

## Intersection Options

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

Intersection Type

Specifies the corridor grade option for the corridors created in the intersection area. Primary Road Crown Maintained: When this option is selected, the primary road crown is maintained through the intersection. All Crowns Maintained: When this option is selected, the crowns of both roads included in the intersection are maintained through the intersection.

Default Exit and Entry Length - Primary Road

Specifies the intersection extents along the primary road as it enters and exits the intersection. If the curb returns are defined, then this parameter is not used, and instead the extents are calculated by the curb return extents on either side of intersection.

Default Exit and Entry Length - Secondary Road

Specifies the intersection extents of the secondary road as it enters and exits the intersection. If the curb returns are defined, then this parameter is not used, and instead the extents are calculated by the curb return extents on either side of intersection.

Create Curb Return Alignments

Specifies whether to create curb return alignments.

Create Curb Return Profiles

Specifies whether to create curb return profiles.

Create Intersection Corridors

Specifies whether to create intersection corridors.

## Default Offsets

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

Primary Road - Left Offset

Specifies the offset distance from the centerline on the left side of the primary road.

**Primary Road - Right Offset**

Specifies the offset distance from the centerline on the right side of the primary road.

**Secondary Road - Left Offset**

Specifies the offset distance from the centerline on the left side of the secondary road.

**Secondary Road - Right Offset**

Specifies the offset distance from the centerline on the right side of the secondary road.

**Offset Length Options**

Specifies how the length of the offset alignment is determined. When To Intersection Extents is selected, offset alignments are extended to the length of the extent of the intersection object. When Alignment Start To End is selected, offset alignments are created the length of the parent alignment.

## Default Cross Slopes

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

**Primary Road - Left Cross Slope**

Specifies the cross slope of the left side of the primary road.

**Primary Road - Right Cross Slope**

Specifies the cross slope of the right side of the primary road.

**Secondary Road - Left Cross Slope**

Specifies the cross slope of the left side of the secondary road.

**Secondary Road - Right Cross Slope**

Specifies the cross slope of the right side of the secondary road.

## Curb Return Parameters

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

**Curb Return Type**

Specifies the type of curb return used for the intersection: Chamfer, Circular Fillet, or 3-Centered Arcs.

**Circular Fillet Radius**

When Circular Fillet is selected as the curb return type, this option lets you specify the radius for the circular fillet on the curb return.

**Radius of Curve (X)**

When 3-Centered Arcs is selected as the curb return type, this option lets you specify the radius for each of the three curves on the curb return.

**Length of Curve (X)**

When 3-Centered Arcs is selected as the curb return type, this option lets you specify the length for the first and third curves on the curb return.

**Chamfer Fillet Length - Along Incoming\Outgoing Lane**

When Chamfer is selected as the curb return type, this option lets you specify the length of the curb return's chamfer fillet along the incoming or outgoing lane.

## Default Widening Parameters

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation. For more information, see Widenings (page 896), or Editing Offset Alignments and Widenings (page 1005).

Widening At Incoming Lane
Specifies whether to widen the incoming lane as it enters the intersection.

Widening At Outgoing Lane
Specifies whether to widen the outgoing lane as it exits the intersection.

Widened Offset
Specifies the offset distance applied to the widened regions, at both the entrance to the intersection, and as it exits the intersection.

Widening Segment Length
Specifies the length of the widened portion of the curb return at both the entrance to the intersection, and as it exits the intersection.

Transition Type
Specifies the type of transition used for the widening: Linear, Curve-Line-Curve, Curve-Curve -Reverse Curve, or Curve - Reverse Curve.

Linear Transition Type
When Linear is selected as the Transition Type, this option lets you specify the type of linear transition used for the widening: By Length or By Taper Ratio.

Transition Length
When By Length is selected as the Linear Transition Type, this option lets you specify the length of the transition for the widening.

Taper Ratio
When By Taper Ratio is selected as the Linear Transition Type, this option lets you specify the ratio of the tapering for the widening.

Radius of Curve X
When a curve option is selected as the Transition Type, this option lets you specify the radius for the first and second curves used in the widening.

Radius of Reverse Curve
When Curve-Curve - Reverse Curve is selected as the Transition Type, this option lets you specify the radius for the first and second curves used in the widening.

## Secondary Road Profile Rules

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

Apply Grade Rules
Specifies whether to apply grade rules.

Maximum Grade
Specifies the maximum grade that can be used in the profile for the secondary road.

Maximum Grade Change
Specifies the maximum grade change that can be used in the profile for the secondary road.

Distance Rule to Adjust the Grade

Specifies the method that is used to adjust the grade of the secondary road profile. When To Intersection Extents is selected, the grade of the secondary road profile is adjusted to within the extents of the intersection object. When Specify Distance is used, the grade of the secondary road profile is adjusted by the value that is entered in the Distance Value.

Distance Value

Specifies the distance that is used to adjust the incoming and outgoing grade along the secondary road profile when Specify Distance is selected for Distance Rule to Adjust the Grade.

## Curb Return Profile Rules

**NOTE** This property group is displayed when accessing the settings from the CreateIntersection command.

Use these settings to specify the default styles assigned to intersection creation.

Define Curb Return Profile By

Specifies the method that is used to define the curb return profile.

Extend Profile Along Incoming Lane

Specifies whether to extend the curb return profile along the incoming lane of the intersection.

Length To Extend Along Incoming Lane

When Extend Profile Along Incoming Lane is set to Yes, this option lets you specify the length to extend the curb return profile along the incoming lane of the intersection.

Extend Profile Along Outgoing Lane

Specifies whether to extend the curb return profile along the outgoing lane of the intersection.

Length To Extend Along Outgoing Lane

When Extend Profile Along Incoming Lane is set to Yes, this option lets you specify the length to extend the curb return profile along the outgoing lane of the intersection.

## Default Styles

Use these settings to specify the default styles assigned to intersection object components.

Intersection Style

Specifies the default style for intersections. Click in the Value column, and click ⋯ to select a style in the Intersection Style dialog box.

Intersection Label Style

Specifies the default style for intersection object labels. Click in the Value column, and click ⋯ to select a label style. Note that there is no placement option available for an intersection label. The label is always placed at the intersection point. If that point is already labeled, then a second label is placed on top of the existing label.

Offset Alignment Style

Specifies the default style for offset alignments. Click in the Value column, and click ⋯ to select a style in the Offset Alignment Style dialog box.

Curb Return Alignment Style

Specifies the default style for curb return alignments. Click in the Value column, and click ⋯ to select a style in the Curb Return Alignment Style dialog box.

Offset Profile Style

Specifies the default style for offset profiles. Click in the Value column, and click ⊡ to select a style in the Offset Profile Style dialog box.

Curb Return Profile Style

Specifies the default style for curb return profiles. Click in the Value column, and click ⊡ to select a style in the Curb Return Profile Style dialog box.

Offset Alignment Label Set

Specifies the default style for offset alignment label sets. Click in the Value column, and click ⊡ to select a style in the Offset Alignment Label Set dialog box.

Curb Return Alignment Label Set

Specifies the default style for curb return alignment label sets. Click in the Value column, and click ⊡ to select a style in the Curb Return Alignment Label Set dialog box.

For general information about style selection, see Select Style Dialog Box (page 1805).

## ⊞ Default Name Format

Use these settings to specify the default name templates used for the following components associated with an intersection object:

■ intersection object

■ intersection quadrant

■ offset alignment

■ curb return alignment

■ offset profile

■ curb return profile

■ corridor region

Click in the Value column, and click ⊡ to select a style in the Name Template dialog box (page 1806).

# Intersection Properties Dialog Box

Use this dialog box to view the properties of an intersection object, such as the object name, description, and style.

The Intersection Properties dialog box also lists details about the intersecting alignments and the corridor regions included in the intersection.

**See also:**

■ Editing Intersection Properties (page 1441)

# Information Tab (Intersection Properties Dialog Box)

Use this tab to view or change general information for the intersection object.

Name

Specifies the name of the current intersection object.

Description

Specifies an optional description for the current intersection.

Object Style

Specifies the default style used to display intersections. Select a style in the list or use the standard style selection tools. For more information about the standard style selection tools, see the Select Style (page 1805) dialog box.

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

# Intersection Details Tab (Intersection Properties Dialog Box)

Use this tab to view or change the parameters of the current intersection object.

Alignment Details

This section displays information about the intersecting alignments in the intersection object. It displays the names of the intersecting alignments, the station of the point of intersection, and the profile associated with each alignment. If there are multiple profiles available, you can change the profile associated with the alignment by clicking in the Profile cell selecting a new profile from the drop-down list, and clicking Apply.

Intersection Quadrants

In this section, you can click Previous and Next to advance through the quadrants in this intersection. For each quadrant, the names of the incoming and outgoing alignments, as well as the intersection angle, are displayed. Most of this information is read-only, however you can change the name of the intersection quadrant if desired.

# Intersection Style Dialog Box

Use this dialog box to make it easier to locate and work with intersection objects by changing their display.

The intersection style controls the appearance of intersection objects.

**See also:**

■ Intersection Styles and Display (page 1425)

# Information Tab (Intersection Style Dialog Box)

Use this tab to change the intersection style name and description information, and to review details about the style, such as when it was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Marker Tab (Intersection Style Dialog Box)

Use this tab to specify the appearance of the marker that is used for intersection objects in the drawing.

**Use AutoCAD POINT For Marker**

Specifies that the assembly component is displayed using the current AutoCAD point symbol, which is specified by the AutoCAD PDMODE and PDSIZE system variables.

**Use Custom Marker**

When this is selected, the specified symbol is used to display the intersection object in the drawing.

Custom Marker Style

Specifies the symbol that is used to display the marker for the intersection object. Click one of the five symbols to the left to use as a base symbol. Optionally, click the sixth symbol, the seventh symbol, or both, to superimpose them over the base symbol.

## Use AutoCAD Block Symbol For Marker

When this is selected, intersection objects are displayed in the drawing using a reference to the selected AutoCAD block. The block is scaled using the options specified in the Size section of this dialog box.

The list displays block definitions available in the drawing. Do one of the following:

- Click a block name to select the block to be used for the assembly symbol.

- Right-click in the block list window and click Browse to select a block located in another folder.

The specified block is displayed in the Preview window.

## Marker Rotation Angle

Specifies the rotation angle for the symbol. Applies to all three symbol types (AutoCAD Point, Custom Marker, and AutoCAD Block). Enter a value or click ⬚ to specify an angle.

## Size

Options

Specifies symbol scaling:

- Use Drawing Scale: Determines size of the marker by multiplying the specified value by the drawing scale. Enter the scale factor.

- Use Fixed Scale: Activates the Fixed Scale options.

- Use Size in Absolute Units: Specifies that the marker size is an absolute value based on the displayed units. Enter the value.

- Use Size Relative to Screen: Specifies that the size of the marker is a percentage of the drawing screen size. Enter the percentage.

Fixed Scale

Specifies independent fixed scale values when Options is set to Use Fixed Scale. Enter values for X, Y, or Z.

## Orient Marker To World Coordinates

Specifies that the maker rotation angle is relative to the world coordinate system.

## Orient Marker To View

Specifies that the maker rotation angle is relative to the current AutoCAD view direction.

# Display Tab (Intersection Style Dialog Box)

Use this tab to change the display and visibility of intersection objects.

## View Direction

Some object styles can have unique display values and varying numbers of displayed components, depending on whether they are being displayed in Plan, Profile, Section, or Model views. You can create and set style characteristics for each supported view direction type using the View Direction list.

Intersection objects support display styles in plan and model view.

Plan and Model

Select Plan or Model to specify display style settings when the intersection object is displayed in plan or model view.

### Component Display

Marker

This is the closed polyline entity that represents the extents of an intersection object. You can set the following style characteristics for this component: Visibility, Layer, Color, Linetype, LT Scale, Lineweight and Plot Style.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Intersection Style Dialog Box)

Use this tab to review general the information about the current intersection object style.

You can copy and paste this information to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Create Intersection Wizard

Use this wizard to create an intersection object after selecting the point where two alignments intersect.

This wizard quickly leads you through the process of creating an intersection. After selecting the point where two alignments intersect, you can configure a variety of parameters related to the intersection using the choices available on the wizard dialog boxes.

Reviewing and changing the settings on each wizard page is optional. From any page of the wizard, clicking Create Intersection creates the intersection using the current values specified.

Before creating an intersection object, make sure you understand the necessary object and drawing data that must exist. See Before You Begin (page 1429) for more information.

Once created, the intersection object is displayed in the drawing and in the Prospector tree. For information about editing intersection objects, see Editing Intersections (page 1439).

## General Page (Create Intersection Wizard)

Use this page of the wizard to specify general details for the intersection object, such as object name and style.

### Intersection Name

Enter a name for the intersection object, or accept the name specified by the default name template for intersection objects. For more information, see Name Template dialog box (page 1806).

### Description

Enter an optional description for this intersection object.

### Intersection Marker Style

Style List

Displays the current style for intersection object markers. Click the arrow to display the intersection marker styles in the drawing.

Style Selection

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box, where you can preview the style and creation information.

## Intersection Marker Layer

Specifies the layer of the intersection object marker. Click  to open the Object Layer dialog box where you can select a different layer for the intersection object marker.

## Intersection Label Style

Style List

Displays the current style for intersection object labels. Click the arrow to display the intersection label styles in the drawing. Note that there is no placement option available for an intersection label. The label is always placed at the intersection point. If that point is already labeled, then a second label is placed on top of the existing label.

Style Selection

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box, where you can preview the style and creation information.

## Intersection Corridor Type

Specifies how crowns will be maintained for this intersection. For more information, see Intersection Corridor Type (page 1434)

Primary Road Crown Maintained

When this option is specified, the crown of the primary road is maintained, while the crown (profile) of the secondary road is adjusted to match the edge of the primary road and the intersection point. The crown (profile and edges) of the primary road are not affected. A conceptual graphic representing this type of intersection is displayed on the wizard dialog box.



All Crowns Maintained

When this option is specified, the profile of the side road is adjusted to match the main road elevation at the intersection point. The main road profile is not affected. The edges are blended together along the curb returns (via the curb return profiles). A conceptual graphic representing this type of intersection is displayed on the wizard dialog box.



**NOTE** Once an intersection object is created, you cannot change the value of the intersection Corridor Type property.

On this dialog box, you can click Next to move to the next wizard dialog box, Create Intersection to create the intersection using the currently selected setting, or Cancel to cancel creating the intersection.

# Geometry Details Page (Create Intersection Wizard)

Use this page of the wizard to specify a variety of details about the geometry of the intersection object.

## Intersecting Alignments

Use this section to view information on the alignments included in this intersection. You can view the alignment name, priority, the station where the intersection occurs, and the profile associated with the alignment. If other profiles exist, you can select a different profile from the drop-down list.

Priority

The priority determines how profile locking occurs for the profiles associated with the intersection. You can change the priority of an alignment (road) during intersection creation by selecting an alignment in this list and using the arrows. It is important to note that when you do this, the associated profiles may change. Once an intersection object is created, you cannot change the value of the priority property. See Changing Road Priority (page 1433) for more information.

## Offset and Curb Returns

Specifies information on the alignment offsets and curb returns for this intersection.

Create Or Specify Offset Alignments

When this option is selected, offset alignments will be created for the intersection. If you do not want offset alignments included in the intersection, remove the check mark from this option. If you do not select an existing alignment from the current drawing, then a new offset alignment object is created. To select an existing offset alignment from the drawing, click the Offset Parameters button, set Use An Existing Alignment to Yes, then select the alignment by clicking ⋯ next to the Alignment Name field.

Offset Parameters

Before deciding to create offset alignments, you can view or edit the parameters that will be used for creating the offset alignments by clicking the Offset Parameters button. This displays the Intersection Offset Parameters dialog box (page 1931) which lets you specify an existing alignment to use, or create a new one according to the offset value you specify. For more information, see Editing Offsets in Intersections (page 1445).

Create Curb Return Alignments

When this option is selected, curb return alignments will be created for the intersection. If you do not want curb return alignments included in the intersection, remove the check mark from this option.

Curb Return Parameters

Before deciding to create curb returns, you can view or edit the parameters that will be used for creating the curb returns by clicking the Curb Return Parameters button. This displays the Intersection Curb Return Parameters dialog box (page 1932), which lets you specify the curb return type (chamfer, circular fillet, or 3-centered arcs), and other details, such as radius for the curb return. For more information, see Editing Curb Returns in Intersections (page 1447).

## Offset and Curb Return Profiles

These options let you create offset and curb return profiles for this intersection. If the intersecting alignments do not have profiles associated with them, these options are not available.

Create Offset and Curb Return Profiles

Selecting this option creates offset and curb return profiles for this intersection. This option is not available when the intersecting alignments selected for this intersection do not have profiles associated with them.

Lane Slope Parameters

Clicking this button displays the Intersection Lane Slope Parameters dialog box where you can view or edit the lane slopes for the roads included in the intersection object. This option is not available when the intersecting alignments selected for this intersection do not have profiles associated with them.

Curb Return Profile Parameters

Clicking this button displays the Intersection Curb Return Profile Parameters dialog box where you can view or edit the parameters for the curb return profile included in the intersection object. This option is not available when the intersecting alignments selected for this intersection do not have profiles associated with them.

On this dialog box, you can click Back to return to previous wizard dialog boxes, Next to move to the next wizard dialog box, Create Intersection to create the intersection using the currently selected setting, or Cancel to cancel creating the intersection.

## Corridor Regions Page (Create Intersection Wizard)

Use this page of the wizard to specify details about the corridor regions included in the intersection object.

### Create Corridors in the Intersection Area

When this option is selected, new corridor objects are created in the intersection area when the intersection is created. When this option is not selected, the rest of the options on this page of the wizard are unavailable. For more information, see Creating Corridors in the Intersection Area (page 1435).

Create a New Corridor

When this option is selected, a new corridor object will be created for this intersection.

Add to an Existing Corridor

When this option is selected, you can select an existing corridor to add to for this intersection. If you do not have corridor objects in the drawing, this option is not available.

Select Surface to Daylight

Specifies a surface to use for daylighting. If you do not have surfaces in the drawing, this option is not available. Also, you cannot target a surface that is an output of another corridor.

### Select Assembly Set to Import

Specifies the assembly set to use for creating the intersection. If desired, you can choose a different assembly set to import for use with creating this intersection. For more information, see Assembly Sets (page 1457).

Browse

Click this button to browse to a location for selecting an assembly set to use for creating this intersection.

Save As Set

Saves the currently selected assembly set to an xml file. When you click this button, you are prompted to choose a name and location for the assembly set.

### Corridor Region Section Type

Specifies the assembly that defines each section (component) in each corridor region included in the intersection. Click the items listed in this section to display a conceptual graphic that updates with each selection.

Assembly to Apply

Displays the name of the assembly that is specified (referenced) in the selected assembly set for each

component. If there are assemblies available in the current drawing, you can click to select a different assembly to apply. If you click an assembly in the list, the location where the assembly will be applied along the intersection is highlighted in the conceptual graphic.

On this dialog box, you can click Back to return to previous wizard dialog boxes, Create Intersection to create the intersection using the currently selected settings, or Cancel to cancel creating the intersection.

When you create an intersection object, the following objects are added to the current drawing:

■ intersection object

■ corridor(s)

■ assemblies

■ subassemblies

■ offset alignments

■ curb return alignments

**Related procedures:**

■ Creating Corridors in the Intersection Area (page 1435)

# Intersection Offset Parameters Dialog Box

Use this dialog box to view and edit the parameters of the offset alignments included in the intersection object.

Click the expand ⬚▾ and collapse ⬚▴ buttons at the top right to expand and collapse the information displayed in this dialog box.

You can select offset alignments for the primary or secondary road in the intersection.

When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location affected by the edit. For more information, see Intersection Editing Visual Cues (page 1443).

### Left or Right Offset Alignment Definition

Specifies the definitions of the offset alignments included in the intersection.

Use an Existing Alignment

When No is specified, the offset alignment is created during intersection object creation. When Yes is specified, an existing alignment from the current drawing can be used to create the left and right alignment offsets.

Offset Value

Specifies the offset distance for the offset alignment. This property is automatically set to Varies if an alignment is selected that is not a dynamic offset alignment based on a centerline.

Alignment Name

Displays the name of the offset alignment currently specified. When Use an Existing Alignment is set to Yes, you can click ⬚ to specify another alignment from the current drawing. When you do this, the Offset Value property is automatically set to Varies, if the selected alignment is not a dynamic offset alignment based on a centerline. It is also important to note that if you do select another offset alignment from the drawing, the original offset alignment that was created during intersection object creation is not deleted.

**Related procedures:**

■ Editing Offsets in Intersections (page 1445)

- Widenings (page 896)
- Editing Offset Alignments and Widenings (page 1005)

# Intersection Curb Return Parameters Dialog Box

Use this dialog box to view and edit the parameters of the curb returns included in the intersection object.

When you select a property item on this dialog box, the affected object, or area of the intersection is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location affected by the edit. For more information, see Intersection Editing Visual Cues (page 1443).

**Previous and Next Buttons**

The Previous and Next buttons let you select each of the quadrants in the intersection sequentially. The currently selected quadrant is highlighted in the drawing.

**Intersection Quadrant**

This drop-down list lets you select each of the quadrants in the intersection.

**Expand and Collapse Buttons**

Click the expand and collapse buttons at the top right to expand and collapse the property information displayed in this dialog box.

**Widen Turn Lane For Incoming or Outgoing Road**

When you select either of these options, the incoming or outgoing road in the intersection is widened.

**Intersection Quadrant Details**

Specifies details for the quadrants in the intersection.

Intersection Quadrant Name

Displays the name of the intersection quadrant currently selected. You can change the name here, if desired.

Incoming Road Centerline Name

Displays the name of the centerline alignment for the incoming road.

Outgoing Road Centerline Name

Displays the name of the centerline alignment for the outgoing road.

Intersection Quadrant Angle

Specifies the angle in degrees that is created by the two alignments that intersect, forming this intersection quadrant.

**Curb Return Parameters**

Specifies parameters for the curb returns in the intersection.

Curb Return Type

Specifies the type of curb return: chamfer, circular fillet, or 3-centered arcs.

Radius

If the radius type is circular fillet or 3-centered arc, you can specify radius parameters.

Length Along Incoming or Outgoing Road

Specifies the length of the curb return for the incoming or outgoing road.

Curve (X) Radius

Specifies the radius for the curves in the curb return.

Curve (X) Length

Specifies the length for each curved section in the curb return.

Copy These To All Quadrants

On this dialog box, you can right-click the "Curb Return Parameters" node in the Property column, and select Copy These To All Quadrants. When this option is selected, the curb return parameters currently selected are automatically applied to all quadrants in the intersection.

### Widening At Incoming\Outgoing Lane

When widening options at the top of this dialog box are selected, you can specify details for the widening at the incoming and outgoing lanes in the intersection.

Transition Details

Specifies the transition details for the incoming or outgoing lanes in the intersection.

Widening Details

Specifies the widening details for the incoming or outgoing lanes in the intersection.

**Related procedures:**

■ Editing Curb Returns in Intersections (page 1447)

## Secondary Road Profile Rules Dialog Box

Use this dialog box to view and edit the parameters of the profile for the secondary roads included in the intersection object.

Click the expand and collapse buttons at the top right to expand and collapse the information displayed in this dialog box.

When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location affected by the edit. For more information, see Intersection Editing Visual Cues (page 1443).

### Secondary Road Profile Rules

Apply Grade Rules

Specifies whether to apply grade rules for this secondary road profile.

Maximum Grade

Specifies the maximum grade that can be used for the secondary road profile in the intersection.

Maximum Grade Change

Specifies the maximum grade change that can be used to adjust the secondary road profile in the intersection.

Distance Rule to Adjust the Grade

Specifies to adjust the grade of the secondary road profile within the intersection extents, or by specifying a distance.

Distance Value

When Distance Rule to Adjust the Grade option is set to Specify Distance, this specifies the distance value used to adjust the incoming and outgoing grade of the secondary road profile in the intersection.

**Related procedures:**

■ Editing Side Road Profiles for Intersections (page 1448)

# Intersection Lane Slope Parameters Dialog Box

Use this dialog box to view and edit the parameters of the lane slopes for the roads included in the intersection object.

Click the expand ⬚▼ and collapse ⬚▲ buttons at the top right to expand and collapse the information displayed in this dialog box.

When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location affected by the edit. For more information, see Intersection Editing Visual Cues (page 1443).

### Left or Right Edge Profile Definition

Specifies the parameters defining the left and right edge of the slope.

Use an Existing Profile

When Yes is selected, you can select a profile from the available profiles in the current drawing. When No is selected, this indicates that the profile was created during intersection creation.

Cross Fall From Centerline

Specifies the slope of the grade from the centerline.

Profile Name

Displays the name of the profile associated with this edge profile definition. When Use An Existing Profile is set to Yes, you can select a new profile from available profiles in the current drawing.

### Related procedures:

■ Editing Lane Slopes for Intersections (page 1449)

# Intersection Curb Return Profile Parameters Dialog Box

Use this dialog box to view and edit the parameters of the curb returns included in the intersection object.

Click the expand ⬚▼ and collapse ⬚▲ buttons at the top right to expand and collapse the information displayed in this dialog box.

When you select an item on this dialog box, the affected object is highlighted in the drawing, and the conceptual graphic on the dialog box updates to indicate the location affected by the edit. For more information, see Intersection Editing Visual Cues (page 1443).

### Intersection Quadrant

This drop-down list lets you specify each quadrant in the intersection. You can also click Next and Previous to advance through the quadrants.

### Incoming\Outoging Lane Details

Specifies curb return profile details for the incoming and outgoing lanes.

Incoming or Outgoing Road Centerline Name

Displays the name of the centerline alignment for the incoming and outgoing road.

Side

Displays which side of the road this profile reflects.

### Curb Return Profile Parameters

Use these settings to view or change the parameters of the curb return profiles included in the intersection creation.

Define Curb Return Profile By

Displays the method that was used to define the curb return profile. For version 2010, the only supported method for defining curb return profiles is by joining tangents.

Extend Profile Along Incoming Lane

Specifies whether to extend the curb return profile along the incoming lane of the intersection.

Length To Extend Along Incoming Lane

When Extend Profile Along Incoming Lane is set to Yes, this option lets you specify the length to extend the curb return profile along the incoming lane of the intersection.

Extend Profile Along Outgoing Lane

Specifies whether to extend the curb return profile along the outgoing lane of the intersection.

Length To Extend Along Outgoing Lane

When Extend Profile Along Incoming Lane is set to Yes, this option lets you specify the length to extend the curb return profile along the outgoing lane of the intersection.

**Related procedures:**

■ Editing Curb Return Profiles for Intersections (page 1450)

# Intersection Corridor Regions Dialog Box

Use this dialog box to view and edit the parameters of the corridor regions included in the intersection object.

**NOTE** If you have made any changes to the corridor in the intersection area, those changes will be lost (overwritten) when you apply new assemblies by clicking Recreate on this dialog box.

For more information, see Creating Corridors in the Intersection Area (page 1435).

Create a New Corridor

Specifies to create a new corridor object to the intersection object.

Add to an Existing Corridor

Specifies to add regions to a corridor object that already exists in the intersection object. If you do not have corridor objects in the intersection drawing, this option is not available.

Select Surface to Daylight

Specifies a surface for daylighting. If there are other available surfaces in the current drawing, you can select a different surface to use for daylighting. If you do not have surfaces in the drawing, this option is not available.

**Apply an Assembly Set**

Specifies an assembly set file that specifies the assemblies used to create the intersection corridors. For more information, see Assembly Sets (page 1457).

Browse

Click this button to browse to and select a new assembly set file.

**Corridor Region Section Type**

Expand the items in this column to display the assemblies that define each section of the corridor region. The left column (Corridor Region Section Type) displays the names of the required sections of the corridor, and is not editable. The right column (Assembly To Apply), displays the names of the assemblies that are currently specified, and you can select different assemblies to apply, if desired.

Assembly to Apply

Displays the names of the assembly that is applied for each section of the corridor region. When you click an assembly in this list, the conceptual graphic on this dialog box displays the location where the assembly is applied in the intersection corridor.

Save As Set

If desired, you can save the set of assemblies currently displayed on this dialog to a new or existing assembly set. For more information, see Assembly Sets (page 1457).

**Related procedures:**

■ Updating Corridor Regions in Intersections (page 1452)

■ Recreating Corridor Regions in Intersections (page 1454)

# Offset Alignment Parameters Dialog Box

Use this dialog box to view and edit the parameters of the currently selected offset alignment.

Click the expand ⊟▼ and collapse ⊟▲ buttons to expand and collapse the region information displayed in this dialog box.

When you select an item on this dialog box, the affected object is highlighted in the drawing. For more information, see Intersection Editing Visual Cues (page 1443).

## Alignment Name

Displays the name of the currently selected offset alignment. Use the drop-down list to select another offset alignment in the drawing.

## Nominal Offset

Specifies the length of the nominal offset for the currently selected offset alignment. If desired, you can change the nominal offset value on this dialog box.

## Add a Widening

Click this button to add widening to the selected offset alignment. Click the button to remove widening.

Region Type

Specifies the type of region this offset alignment is associated with. Use the drop-down list to specify Nominal or Specific.

Start Station

Specifies the start station for the currently selected offset alignment. You can specify a different station by entering a value, or by clicking a station along this alignment in the drawing.

End Station

Specifies the end station for the currently selected offset alignment. You can specify a different station by entering a value, or by clicking a station along this alignment in the drawing.

Region Length

Specifies the length of the region this offset alignment is associated with.

**Related procedures:**

■ Widenings (page 896)

■ Editing Offset Alignments and Widenings (page 1005)

# Label Dialog Boxes

# 55

Use the following links to access information about Label dialog boxes.

## Standard Edit Label Style Defaults Dialog Box

The same standard settings are available in each of the Edit Label Style Defaults dialog boxes. See The Hierarchy of Label Settings (page 1468) for information about how each level of setting affects the others.

### Label

Text Style

Specifies a default text style for all label text components in a drawing. The default is Standard. Select a text style from the list of all AutoCAD text styles defined in the current drawing.

Visibility

Specifies whether all label styles in a drawing are visible.

---

**TIP** Use this setting to quickly turn all labels in the drawing off or on.

---

Layer

Specifies the default layer for all label styles in a drawing.
The default layer is 0. When the layer is set to 0, the labels use the properties of the parent object layer specified on the Object Layers tab of the Drawing Settings dialog box.

Click  to display the Layer Selection dialog box.

---

**NOTE** This setting specifies the layer from which the labels obtain their display properties. The labels are drawn on the parent object layer.

---

### Behavior

Orientation Reference

Specifies the orientation reference of the labels.

■ **Object**: Rotates labels relative to the zero direction of the object. You can determine the zero direction of the object based on its start and end points. If the object vector changes at the anchor point on the label, the orientation updates automatically. This is the default setting.

■ **View**: Forces labels to realign to a screen-view orientation in both model and layout views. Always assumes the zero angle is horizontal, regardless of UCS or Dview twist. If the view changes, the label orientation updates with it.

- **World Coordinate System**: Adjusts the labels with respect to the angle between the current view and world view. Changing the view or current UCS does not affect label rotation with respect to the world coordinate system.

For more information, see Changing Label Orientation (page 1488).

Forced Insertion

Specifies the position of a label relative to an object. Applies only when the Orientation Reference option is set to Object and the objects are lines, arc segments, or spline segments.

- **None**: Maintains label position as composed relative to the object. This is the default setting.

- **Top**: Adjusts label position to above an object.

- **Bottom**: Adjusts label position to below an object.

**NOTE** Plan Readable should be set to True when using the Top or Bottom settings.

Force Inside Curve

Specifies whether labels are placed inside or outside a curve. Available only for curve label styles. This setting has precedence over any forced insertion setting.

- **True**: Moves label components on the outside of curve to the inside of curve. The components maintain the same offset and rotation.

- **False**: All label components remain as composed. This is the default setting.

**NOTE** These settings apply only when the Orientation Reference option is set to Object.

## Plan Readability

Plan Readable

Specifies the text rotation to insure that all text components in labels can be read easily in plan view.

- **True**: Rotates text to insure that it can be read easily in plan view or as if viewed from an angle at the bottom or right side of the screen/paper. This is the default setting.
  Any text with an angle greater than the angle specified in the Readability Bias setting, or with an angle less than the Readability Bias plus 180 degrees, is in violation of plan readability, and is adjusted automatically.

  For more information about plan-readable text, see Using Plan-Readability (page 1489).

- **False**: Displays text as inserted.

This option applies only to text components in labels.

Readability Bias

Specifies the angle at which label text flips 180 degrees to remain plan readable.

Flip Anchors With Text

- True: Ensures that if text is rotated to make plan-readable, the anchors will also be flipped.

- False: The flipped label always looks like a mirror of the original.

## Components

Text Height

Specifies the plotted height for label text. Enter a positive value greater than zero. The value is applied to all text components.

Color

Specifies the color for label components. Click  to open the Select Color dialog box.

Linetype

Specifies the linetype for label components. Click  to open the Select Linetype dialog box.

Lineweight

Specifies the lineweight for label components. Click  to open the Lineweight dialog box.

Span Outside Segments

Specifies whether the label style component should span outside segments.

**NOTE** This setting applies to parcel line and curve label styles only).

- **True**: Labels the outer boundary of parcels, rather than the individual parcel segments. For example, if four parcels share an outer boundary, use this option to label the combined outer boundary.

- **False**: Labels individual parcel segments.

## Leader

Arrow Head Style

Specifies an arrow head style for a leader.
Select an arrow head style from the list. Or, select None to display leaders without arrow heads. The default arrow head style is Closed Filled.
For more information about arrow head styles, see "Choose Dimension Arrowheads" in AutoCAD Help.

Arrow Head Size

Specifies the size of the arrow head. The default arrow head size is defined in plot units (either inches or millimeters). The plot units are determined by the drawing units established in the drawing settings.
Enter zero or a positive number. A value of zero means that a leader is displayed without an arrow head.

Visibility

Specifies whether leaders are visible when you drag a label from its original position.

Type

Specifies the shape of the leader.

- **Straight Leader**: Draws a straight leader when you drag a label. This is the default leader type.

- **Spline Leader**: Draws a spline (or smooth curve) leader.

Color

Specifies the color for leaders. Click  to open the Select Color dialog box.

Linetype

Specifies the linetype for leaders. Click  to open the Select Linetype dialog box.

Lineweight

Specifies the lineweight for leaders. Click  to open the Lineweight dialog box.

**Dragged State Components**

Display

Specifies how label content is displayed after it is dragged from its default position.

- **As Composed**: Maintains the original settings for orientation and composition. When selected, all other properties in the Dragged State Components category are unavailable for editing. This is the default.

- **Stacked Text**. Reformats labels based on the settings in this Dragged State Components category.

---

**NOTE** When Display is set to Stacked Text, all blocks, lines, and direction arrows are removed. The text components are stacked vertically based on the order in which the text components were created in the label style.

---

Border Visibility

Specifies whether a border is visible for dragged labels.

Border Type

Specifies the shape of the border.

- **Rectangular**: Draws a rectangle.

- **Rounded Rectangular**: Draws a rectangle with rounded corners.
  The radius used to create a rounded rectangle is calculated by adding the border and leader gap value and half the overall text height (including descending characters, and subscript and superscript characters).

- **Circular**: Draws a circular border.

Border and Leader Gap

Specifies the distance between the leader and the label text. Enter a positive value greater than zero. The gap value is also used to define the space between the border and the label text.

Text Height

Specifies the plotted height for all text components. Enter a positive value greater than zero.

---

**NOTE** When Display is set to As Composed, this property is unavailable for editing.

---

Leader Attachment

Specifies the location where a leader hook is drawn in relation to label content.

- Top Of Top Line (of multiple lines of text)

- Middle Of Top Line (of multiple lines of text)

- Middle (of single line of text)

- Middle Of Bottom Line (of multiple lines of text)

- Bottom Of Bottom Line (of multiple lines of text)

Middle is the default setting.

Leader Justification

Specifies how label text is justified in relation to the leader.

- **True**. Text is left-justified when the leader is on the left, and right-justified when the leader is on the right, as shown in the following illustration. This is the default setting.

■ **False**. Text is left-justified regardless of leader location, as shown in the following illustration:



Color

Specifies the color for dragged components. Click [icon] to open the Select Color dialog box.

Linetype

Specifies the linetype for dragged components. Click [icon] to open the Select Linetype dialog box.

Lineweight

Specifies the lineweight for dragged components. Click [icon] to open the Lineweight dialog box.

[icon] Collapse All Categories

Collapses the property categories to show only the top-level item.

[icon] Expand All Categories

Expands categories to show all properties.

[icon] Override All Dependencies

Selects all the check boxes in the Override column, which prevents the setting from being changed if the value is changed at a higher level setting. This option is not available at the drawing level.

# Edit Label Style Defaults Dialog Box (Drawing Level)

Use this dialog box to set default settings for all labels in the current drawing.

For a description of the controls, see Standard Edit Label Style Defaults Dialog Box (page 1937).

The defaults that you set in this dialog box are global until overridden by changes made to default settings at the object collection level (such as Point, Parcel, Surface, and Alignment); at the label-style-type-level; or at the label style level. For a description of override controls unique to managing the hierarchy of default label settings, see Managing Overrides in Label Style Dialog Boxes (page 1960).

**Related procedures:**

■ Default Settings for All Label Styles in a Drawing (page 1475)

# Edit Label Style Defaults Dialog Box (Feature Level)

Use this dialog box to set default settings for all label styles for an object type.

For a description of the controls, see Standard Edit Label Style Defaults Dialog Box (page 1937).

These default settings can be dependent on the higher default settings at the drawing level (page 1941), or you can override the higher settings by clicking the property and entering a value. Editing the property places a check mark in the Override column, and the changes you make are applied to lower-level settings and in individual label styles. For a description of override controls that are unique to managing the hierarchy of default label settings, see Managing Overrides in Label Style Dialog Boxes (page 1960).

**Related procedures:**

■ Default Settings for All Label Styles in a Feature (page 1475)

# Edit Label Style Defaults Dialog Box (Label Style Type Level)

Use this dialog box to define default settings for all labels belonging to a specific label style type.

For a description of the controls, see Standard Edit Label Style Defaults Dialog Box (page 1937).

These default settings can be dependent on higher settings at the drawing level (page 1941) or at the feature (page 1941) level.

You can override those settings by clicking the property and entering another value. For a description of override controls unique to managing the hierarchy of default label settings, see Managing Overrides in Label Style Dialog Boxes (page 1960).

**Related procedures:**

■ Default Settings for All Labels in a Label Type (page 1476)

# Label Style Composer Dialog Box

Use this dialog box to create new label styles or edit both the format and content of existing label styles.

**NOTE** The controls and settings available in the dialog box vary for each object type.

## Information Tab (Label Style Composer Dialog Box)

Use the Information tab to change the label style name and description information, and to review details, such as when the style was most recently modifed.

Name

Specifies a name for the current style.

**NOTE** Label styles names are case sensitive. "Standard" is not the same as "standard".

Description

Specifies a description of the current style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the date and time the style was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the date and time the style was last modified.

## General Tab (Label Style Composer Dialog Box)

Use this tab to specify whether the label displays as a label or tag, the label visibility, and label orientation.

Property column

Lists categories and label properties. By clicking +/- next to a category name, you can expand or collapse the list of label properties for that category.

When a lock 🔒 is displayed to the left of the property name, the property's value has been locked at a higher level setting and all lower settings are also locked.

Value column

Specifies the current value of the property. The value is either derived from a dependency on a higher setting or it can be set explicitly at the current level.

## Label

Text Style

Specifies the default text style on which all text components are based.

Visibility

Specifies whether labels of the current style are visible in the drawing.

Display Mode

Specifies whether labels are displayed as labels or tags.

- **Label**: Displays label components that are assigned either the Label Mode or the Label And Tag Modes "Used In" setting on the Layout tab.

- **Tag**: Displays label components that are assigned either the Tag Mode or the Label and Tag Modes "Used In" setting on the Layout tab.

---

**NOTE** Only the label styles that support tables contain this property.

---

Layer

Specifies the layer for all components in a label style.

Click ⋯ to open the Layer Selection dialog box, and select a layer.
If you set the layer to 0, the labels use the properties of the parent object layer specified on the Object Layers tab of the Drawing Settings dialog box.

---

**NOTE** There are two options for the style to reference a layer for display control, when using ByBlock or ByLayer controls. Reference the layer as specified in the style or reference the layer that the the object resides on (by setting the Layer to 0 in the style. If layer 0 is selected, any ByBlock and ByLayer components will display according to the actual layer that the layer object is residing on in AutoCAD. If a layer other than 0 is selected, any ByBlock or ByLayer components will display according to the layer as specified in the style.

---

## Behavior

Orientation Reference

Specifies the orientation reference of a label.

- **Object**: Rotates labels relative to the zero direction of the object. You can determine the zero direction of the object based on its start and end points. If the object vector changes at the anchor point on the label, the orientation updates automatically. This is the default setting.

- **View**: Forces labels to realign to a screen-view orientation in both model and layout views. Always assumes the zero angle is horizontal, regardless of UCS or Dview twist. If the view changes, the label orientation updates with it.

- **World Coordinate System**: Adjusts the labels with respect to the angle between the current view and world view. Changing the view or current UCS does not affect label rotation with respect to the world coordinate system.

**Forced Insertion**

Specifies the position of a label relative to an object. Applies only when the Orientation Reference option is set to Object and the objects are lines, arc segments, or spline segments.

- **None**: Maintains label position as composed relative to the object.

- **Top**: Adjusts label position to above an object.

- **Bottom**: Adjusts label position to below an object.

> **NOTE** Plan Readable should be set to True when using the Top or Bottom settings.

**Force Inside Curve**

Specifies whether labels are placed inside or outside a curve. Available only for curve label styles. This setting has precedence over any forced insertion setting.

- **True**: Moves label components on the outside of curve to the inside of curve. The components maintain the same offset and rotation.

- **False**: All label components remain as composed.

> **NOTE** This setting applies only when the Orientation Reference option is set to Object.

### Plan Readability

**Plan Readable**

Specifies the text rotation to insure that all text components in labels can be read easily in plan view.

- **True**: Rotates text to insure that it can be read easily in plan view or as if viewed from an angle at the bottom or right side of the screen/paper.
  Any text with an angle greater than the angle specified in the Readability Bias setting, or with an angle less than the Readability Bias plus 180 degrees, is in violation of plan readability, and is adjusted automatically.

- **False**: Displays text as inserted.

**Readability Bias**

Specifies the angle at which label text flips 180 degrees to remain plan readable.

**Flip Anchors With Text**

- True: Ensures that if text is rotated to make plan-readable, the anchors will also be flipped.

- False: The flipped label always looks like a mirror of the original.

### Preview Pane

Dynamically displays changes to the label style. Right-click the preview pane to access view-related commands. To change the view, select a preview drawing from the list. For more information, see Previewing Label Styles (page 1472).

**Related procedures:**

- Managing General Properties for Label Styles (page 1486)

## Layout Tab (Label Style Composer Dialog Box)

Use this tab to create and edit label components.

See the following topics to access information about defining layout properties for each label component.

**See also:**

■ Managing Layout Properties for Label Styles (page 1492)

## Common Elements of the Layout Tab

Use controls at the top of the Layout tab to select, create, copy, delete, or change the draw order of label components.

Component Name

Displays a list of existing label components defined for the label style. Select a component from this list to edit its properties.

Create Component

Creates a new label component. Select a component type from this list to create a new text, text for each, block, direction arrow, tick, or line component. The Property and Value columns display settings you can change to define the appearance, location, and orientation of that particular component. Not all components are available for all label types.

Copy Component

Copies the selected component.

Delete Component

Deletes the selected component.

Change Draw Order

Displays the Component Draw Order (page 1958) dialog box.

Preview Pane

Dynamically displays changes to the label style. Right-click the preview pane to access view-related commands. To change the view, select a preview drawing from the list. For more information, see Previewing Label Styles (page 1472).

Color

Specifies colors for components, borders, and leaders.

Click the Value column, and then click to open the Select Color dialog box.

Linetype

Specifies linetypes for components and borders.

Click the Value column, and then click to open the Select Linetype dialog box.

Lineweight

Specifies the lineweights for components and borders.

Click the Value column, and then click to open the Lineweight dialog box.

## Text Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a text component for a label style.

From the Create Component list , click to create a new text component.

## General

Name

Specifies the name of the text component.

The default name, such as "Text.1," is the component name with a numeric increment. If the text component exists in a parent label style, then the name cannot be edited.

Visibility

Specifies whether the text component is visible in the label style.

Used In

Specifies whether the component is displayed in Tag Mode, Label Mode, or both. Select a mode on the General tab (page 1942) in the Label Style Composer.

- **Label Mode**: Displays the text component when Display Mode is set to Label.

- **Tag Mode**: Displays the text component when Display Mode is set to Tag.

- **Label and Tag Modes**: Displays the text component regardless of the display mode setting.

**NOTE** If a label style type does not support tables, then this control is not available.

Anchor Component

Specifies a reference object for positioning the text component. You can select <Feature> (which is the object being labeled) or another existing label component.

Anchor Point

Specifies the location on the Anchor Component where the text component is attached.

- When <Feature> is the Anchor Component, Label Location is the only option available. This option places the anchor point at the location where the label is attached to the object.

- When the Anchor Component is another label component, you have a choice of anchor points depending on whether the anchor is a text, block, tick, line, or direction arrow component. For example, if the anchor is a text component, you can select from the points shown in the following illustration:



Span Outside Segments

Specifies whether the label style component should span outside segments.

**NOTE** This setting applies to parcel line and curve label styles only.

- **True**: Labels the outer boundary of parcels, rather than the individual parcel segments. For example, if four parcels share an outer boundary, use this option to label the combined outer boundary.

- **False**: Labels individual parcel segments.

## Text

Contents

Specifies the content of the text component. When a text component is first created, displays "Label Text" by default.

Click the Value column, and then click ⌷ to open the Text Component Editor (page 1961) dialog box where you can create and edit the label content.

Text Height

Specifies the plotted height for text. Enter a positive value greater than zero.

Rotation Angle

Specifies the angle for the text component. Enter a positive or negative value, or click to select an angle in the drawing.

The angle direction is always counterclockwise, and the zero (0) direction is determined by the anchor component type.

If the text component is anchored to a feature, then the zero direction is determined in relation to the following Orientation Reference settings on the General (page 1942) tab:

- **Object**: Calculates the zero angle by examining the object's construction and then measuring the zero angle from the start to the end of the object. If the object is a curve, then the zero angle is tangent to the curve.

- **View**: Sets the zero angle equal to the world coordinate system (WCS) base angle of East (horizontal, left to right).

- **World Coordinate System:** Sets the zero angle equal to the WCS base angle of East (horizontal, left to right).

| If the text component is anchored to... | then the zero angle is measured... |
|---|---|
| another text component, a block component, tick component | along the component's zero angle. |
| line component or vertex arrow component | from the start of the line to the end of the line. |



Attachment

Specifies the location on the text component that is attached to the anchor point. Attachment points are calculated based on a rectangle that encompasses the text, the size of which is determined by including the Gap value specified in the Border category. The following illustration shows text attachment points:

X Offset

Specifies the offset distance between the attachment point and the anchor point in the X direction (zero angle direction).

The X direction is determined by the object to which the text component is anchored. If the text component is anchored to the label insertion point, then the X direction is determined in relation to the following Orientation Reference settings on the General (page 1942) tab:

- **Object**: Calculates the X direction by examining the object's construction and measuring the zero angle direction from the start to the end of the object. If the object is a curve, then the zero angle is located tangent to the curve.

- **View**: The X direction is always horizontal, left to right, regardless of the User Coordinate System (UCS) or Dview twist direction.

- **World Coordinate System**: The X direction is the same as the User Coordinate System (UCS) base angle of East (horizontal, left to right).

Y Offset

Specifies the offset distance between the attachment point and the anchor point in the Y direction, which is 90 degrees counterclockwise to the X direction.

Allow Curved Text

Specifies whether text is drawn along a curve for curve label styles.

## Border

Visibility

Specifies whether the border is visible.

Type

Specifies the shape of the border.

- **Rectangular**: Draws a rectangle around the text component.

- **Rounded Rectangular**: Draws a rectangle with round corners around the text component.
  The radius used to create a rounded rectangle is calculated by adding the gap value and half the overall text height (including descending, subscript, and superscript characters).

Gap

Specifies the distance between the border and text. The Gap is still applied if Border Visibility is set to False.

Background Mask

Specifies whether a mask is applied to the component using the border shape and size.

without mask



1234

with mask



1234

**Related procedures:**

■ Text (page 1502)

■ Managing Layout Properties for Label Styles (page 1492)

## Text-For-Each Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a Text-For-Each component for a label style.

Text-For-Each components label each object associated with a parent object (the object you are labeling). For example, you can use a Text-For-Each component in a structure label style to include labels for pipes associated with that structure. For more information, see Text-For-Each (page 1511).

From the Create Component list  , click  to create a new Text-For-Each component.

The Select Type (page 1949) dialog box is displayed, where you select the type of data to include in the Text-For-Each component.

**NOTE** The Text-For-Each component type is available only for structure label styles.

After you select the Text-For-Each type, your selection is displayed on the Layout tab in the General section.

The remainder of the settings that you specify for a Text-For-Each component are identical to the settings for a text component. For more information, see Text Component Layout (Label Style Composer Dialog Box) (page 1945).

## Select Type Dialog Box

Use the Select Type dialog box to select the type of Referenced Text or Text-For-Each label component to create.

### Referenced Text Options

Alignment
   Creates a Referenced Text component with property fields you can use to refer to alignment data.

Profile
   Creates a Referenced Text component with property fields you can use to refer to profile data.

Surface
   Creates a Referenced Text component with property fields you can use to refer to surface data.

Parcel
   Creates a Referenced Text component with property fields you can use to refer to parcel data.

**Text-For-Each Options**

Structure All Pipes

When inserted into a structure label style, adds labels for all pipes coming into or out of the structure.

Structure In Flow Pipes

When inserted into a structure label style, adds labels for all pipes coming into the structure.

Structure Out Flow Pipes

When inserted into a structure label style, adds labels for all pipes going out of the structure.

## Referenced Text Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a Reference Text component for a label style.

Reference Text components refer to other objects in the drawing, instead of to the object you are labeling.

From the Create Component list ![A icon] , click ![A icon] to create a new Referenced Text component.

The Select Type (page 1949) dialog box is displayed, where you select the type of data to include in the Referenced Text component.

**NOTE** The Referenced Text component type is available for all new label objects. See Label Objects (page 1463)for a list of supported label objects.

After you select the Referenced Text type, the object type you selected is displayed on the Layout tab in the General section.

**NOTE** Once you specify the object type, it cannot be changed. If you want to insert references to a different object type, you must create a new Referenced Text component and select the other object type when you create the new component.

The remainder of the settings that you specify for a Referenced Text component are identical to the settings for a text component. For more information, see Text Component Layout (Label Style Composer Dialog Box) (page 1945). However, when you set up the label contents, you select Properties that refer to the referenced object.

**Related procedures:**

■ Creating Note Labels (page 1480)

■ Inserting Note Labels (page 1530)

## Block Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a block component for a label style.

From the Create Component list ![A icon] , click ![block icon] to create a new block component.

**General**

Name

Specifies the name of the block component.
The default name, such as "Block.1," is the component name with a numeric increment. If the block component exists in a parent label style, then the name cannot be edited.

**Visibility**

Specifies whether the block component is visible in the label style.

**Used In**

Specifies whether the component is visible in tag mode, label mode, or both. Select a mode on the General tab (page 1942) in the Label Style Composer.

- **Label Mode**: Displays the block component when Display Mode (on the General tab) is set to Label.

- **Tag Mode**: Displays the block component when Display Mode is set to Tag.

- **Label and Tag Modes**: Displays the block component regardless of the display mode setting.

---

**NOTE** If a label style type does not support tables, then this control is not available.

---

**Anchor Component**

Specifies a reference object for positioning the block component. You can select <Feature> (which is the object being labeled) or another existing label component.

**Anchor Point**

Specifies the location on the Anchor Component where the block component is attached.

- When <Feature> is the Anchor Component, Label Location is the only option available. This option places the anchor point at the location where the label is attached to the object.

- When the Anchor Component is another label component, you have a choice of anchor points depending on whether the anchor is a text, block, tick, line, or direction arrow component.

## Block

**Block Name**

Specifies the block to use in the block component.

Click the Value column, and then click ⊡ to open the Select A Block dialog box.

In the Select A Block dialog box, click 🔍 to open a Viewer in which you can use standard AutoCAD viewing tools to preview the block.

---

**NOTE** The block must exist in the current drawing.

---

**Block Height**

Specifies the height to which the block is scaled to fit. The block's extents in the X direction are used for scaling the height. The block's aspect ratio is maintained when scaled.



**Rotation Angle**

Specifies the angle for the block component. Enter a positive or negative value, or click to select an angle in the drawing.
The angle direction is always counterclockwise, and the zero (0) direction is determined by the anchor component type.

Attachment

Specifies the location on the block component that is attached to the anchor point. Attachment points are calculated based on a rectangle that encompasses the block, the size of which is determined by including the Gap value specified in the Border category. The following illustration shows block attachment points:



X Offset

Specifies the offset distance between the attachment point and the anchor point in the X direction (zero angle direction).

The X direction is determined by the object to which the block component is anchored. If the block component is anchored to the label insertion point, then the X direction is determined in relation to the following Orientation Reference setting on the General (page 1942) tab:

- **Object**: Calculates the X direction by examining the object's construction and measuring the zero angle direction from the start to the end of the object. If the object is a curve, then the zero angle is located tangent to the curve.

- **View**: The X direction is always horizontal, left to right, regardless of the User Coordinate System (UCS) or Dview twist direction.

- **World Coordinate System**: The X direction is the same as the User Coordinate System (UCS) base angle of East (horizontal, left to right).

Y Offset

Specifies the offset distance between the attachment point and the anchor point in the Y direction, which is 90 degrees counterclockwise to the X direction.

**Related procedures:**

- Blocks (page 1513)

- Managing Layout Properties for Label Styles (page 1492)


## Line Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a line component for a label style.

From the Create Component list , click to create a new line component.

A line component can be defined in two ways, depending on the Use End Point Anchor setting:

- **Start Point, Length, and Angle**: Use this method to anchor the start point of the line component to the anchor component. Set the Use End Point Anchor setting to False, and then specify the Length and Angle.

- **Start Point and End Point**: Use this method to anchor the start and end point of the line component. Set the Use End Point Anchor setting to True, and then specify the End Point Anchor Component and End Point Anchor Point. The length and angle of the line are determined by the position of the anchor objects, and adjust if the anchor components are edited.

## General

Name

Specifies the name of the line component.
The default name, such as "Line.1," is the component name with a numeric increment. If the line component exists in a parent label style, then the name cannot be edited.

Visibility

Specifies whether the line component is visible in the label style.

Used In

Specifies whether the component is visible in tag mode, label mode, or both. Select a mode on the General tab (page 1942) in the Label Style Composer.

- **Label Mode**: Displays the line component when Display Mode is set to Label.

- **Tag Mode**: Displays the line component when Display Mode is set to Tag.

- **Label and Tag Modes**: Displays the line component regardless of the display mode setting.

**NOTE** If a label style type does not support tables, then this control is not available.

Start Point Anchor Component

Specifies a reference object for positioning the start point of the line component. You can select <Feature> (which is the object being labeled) or another existing label component.

Start Point Anchor Point

Specifies the location on the Start Point Anchor Component where the start point of the line component is attached.

- When <Feature> is the Anchor Component, Label Location is the only option available. This option places the anchor point at the point where the label is attached to the object being labeled.

- When another label component is the Anchor Component, you have a choice of anchor points depending on whether the anchor is a text, block, tick, line, or direction arrow component.

Use End Point Anchor

Specifies the method for defining the line component.

- **True**: Uses the Start Point and End Point method to define the line component.
  The Length and Angle properties in the Line category become unavailable for editing.

- **False**: Uses the Start Point, Length, and Angle method to define the line component.
  The End Point Anchor Component, End Point Anchor Point, End Point X Offset, and End Point Y Offset properties become unavailable for editing.

End Point Anchor Component

Specifies a reference object for positioning the end point of the line component.
This property is unavailable when the Use End Point Anchor property is set to False.

End Point Anchor Point

Specifies the location on the End Point Anchor Component where the end point of the line component is attached.
This property is unavailable when the Use End Point Anchor property is set to False.

## Line

Length

Specifies the length of the line component.
This property is unavailable when the Use End Point Anchor property is set to True.

Angle

Specifies the angle of the line component from the start point. Enter a positive or negative value, or click to select an angle in the drawing.

This property is unavailable when the Use End Point Anchor property is set to True.

Start Point X Offset

Offsets the line start point from the start point anchor point in the X direction.

The X direction is determined by the object to which the line component is anchored. If the line component is anchored to the label insertion point, then the X direction is determined in relation to the following Orientation Reference settings on the General (page 1942) tab:

- **Object**: Calculates the X direction by examining the object's construction and measuring the zero angle direction from the start to the end of the object. If the object is a curve, then the zero angle is tangent to the curve.

- **View**: The X direction is always horizontal, left to right, regardless of the User Coordinate System (UCS) or Dview twist direction.

- **World Coordinate System**: The X direction is the same as the User Coordinate System (UCS) base angle of East (horizontal, left to right).

Start Point Y Offset

Offsets the line start point from the start point anchor point in the Y direction.

End Point X Offset

Offsets the line end point from the end point anchor point in the X direction.

This property is unavailable when the Use End Point Anchor property is set to False.

End Point Y Offset

Offsets the line end point from the end point anchor point in the Y direction.

This property is unavailable when the Use End Point Anchor property is set to False.

**Related procedures:**

- Lines (page 1512)

- Managing Layout Properties for Label Styles (page 1492)

## Tick Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a tick component for a label style.

From the Create Component list , click  to create a new tick component.

**NOTE** Unlike other label components, ticks are always anchored to the feature that is being labeled. Therefore ticks do not have Anchor Component or Anchor Point properties.

### General

Name

Specifies the name of the tick component.

The default name, such as "Tick.1," is the component name with a numeric increment. If the tick component exists in a parent label style, then the name cannot be edited.

Visibility

Specifies whether the tick component is visible in the label style.

Used In

Specifies whether the component is visible in Tag Mode, Label Mode, or both. Select a mode on the General tab (page 1942) in the Label Style Composer.

- **Label Mode**: Displays the tick component when Display Mode is set to Label.

- **Tag Mode**: Displays the tick component when Display Mode is set to Tag.

- **Label and Tag Modes**: Displays the tick component regardless of the display mode setting.

**NOTE** If a label style type does not support tables, then this control is not available.

## Tick

Block Name

Specifies the block to use for the tick component.

Click the Value column, and then click ⬚ to open the Select A Block dialog box.

In the Select A Block dialog box, click 🔍 to open a Viewer in which you can use standard AutoCAD viewing tools to preview the block.

**NOTE** The block must exist in the current drawing.

Block Height

Specifies the height to which the tick block is scaled to fit. The block's extents in the X direction are used for scaling the height. The block's aspect ratio is maintained when scaled.



Rotation Angle

Specifies the angle for the tick component. Enter a positive or negative value, or click to select an angle in the drawing.

Align With Object

Specifies whether the tick is inserted at an angle relative to the object.

- **True**: Inserts the tick at an angle relative to the object:



- **False**: Inserts the tick at an angle relative to the UCS setting:

**NOTE** The specified Rotation Angle is applied in conjunction with this setting.

**Related procedures:**

- Ticks (page 1514)

- Managing Layout Properties for Label Styles (page 1492)


# Direction Arrow Component Layout (Label Style Composer Dialog Box)

Use the Layout tab to create a direction arrow component for a label style.

From the Create Component list , click to create a new direction arrow component.

### General

Name
> Specifies the name of the direction arrow component.
> The default name, such as "Direction Arrow.1," is the component name with a numeric increment. If the arrow component exists in a parent label style, then the name cannot be edited.

Visibility
> Specifies whether the direction arrow component is visible in the label style.

Used In
> Specifies whether the component is visible in tag mode, label mode, or both. Select a mode on the General tab (page 1942) in the Label Style Composer.

> - **Label Mode**: Displays the direction arrow component when Display Mode is set to Label.

> - **Tag Mode**: Displays the direction arrow component when Display Mode is set to Tag.

> - **Label and Tag Modes**: Displays the direction arrow component regardless of the display mode setting.

> **NOTE** If a label style type does not support tables, then this control is not available.

Anchor Component
> Specifies a reference object for positioning the direction arrow component. You can select <Feature> (which is the object being labeled) or another existing label component.

Anchor Point
> Specifies the location on the Anchor Component where the direction arrow component is attached.

> - When <Feature> is the Anchor Component, Label Location is the only option available. This option places the anchor point at the point where the label is attached to the object being labeled.

> - When the Anchor Component is another label component, you have a choice of anchor points depending on whether the anchor is a text, block, tick, line, or direction arrow component.

Span Outside Segments
> Specifies whether the label style component should span outside segments.

> **NOTE** This setting applies to parcel line and curve label styles only).

> - **True**: Labels the outer boundary of parcels, rather than the individual parcel segments. For example, if four parcels share an outer boundary, use this option to label the combined outer boundary.

> - **False**: Labels individual parcel segments.

**Direction Arrow**

Arrow Head Style

Specifies the arrow head style. Select an arrow head style from the list.

**NOTE** Select None to create a direction arrow without an arrow head.

Arrow Head Size

Specifies the size of the arrow head in plot units.

**NOTE** A value of zero creates a direction arrow without an arrow head.

**NOTE** The direction arrow length must be at least twice that of the specified arrow head size. If not, the arrow head is not displayed. For example, if you specify an arrow head size of 0.2, the arrow length must be at least 0.4. This behavior is similar to AutoCAD leaders.

**Arrow head is not displayed when arrow length is decreased**

Fixed Length

Specifies whether the arrow length is controlled by the Length setting or by the length of the object being labeled.

■ **True**: Uses the Length setting.

■ **False**: Draws the length of the direction arrow to match the length of the object being labeled.

Length

Specifies the overall length of the direction arrow, including the arrow head.
The Length setting is unavailable when the Fixed Length property is set to False.

X Offset

Specifies the offset distance between the mid-point of the direction arrow and the anchor component in the X direction.

Y Offset

Specifies the offset in the Y direction.
The following illustration shows X and Y offsets for a direction arrow:

**Related procedures:**

■ Direction Arrows (page 1515)

■ Managing Layout Properties for Label Styles (page 1492)

## Component Draw Order Dialog Box

Use this dialog box to control how label text is drawn. Move a component up in the display order to ensure it appears on top of other components.

Component List

Lists the defined components in their current draw order. The component at the top is drawn last. Click a component name and then click the Top or Bottom arrows to move the component to a different draw order location.

# Dragged State Tab (Label Style Composer Dialog Box)

Use this tab to define properties for labels when they are dragged away from their insertion points.

### Leader

Arrow Head Style

Specifies the arrow head style for the leader. Select an arrow head style from the list.

**NOTE** Select None to create a leader without an arrow head.

Arrow Head Size

Specifies the size of the arrow head in plot units.

**NOTE** A value of zero creates a leader without an arrow head.

Visibility

Specifies whether the leader is visible when a label is dragged from its default position.

Type

Specifies the shape of the leader.

■ **Straight Leader**: Draws a straight leader.

■ **Spline Leader**: Draws a spline leader.

### Dragged State Components

Display

Specifies how label content is displayed after it is dragged from its default position.

■ **As Composed**: Labels maintain the original settings for composition and orientation.
When you select As Composed, all other properties in the Dragged State Components category become unavailable for editing.

■ **Stacked Text**: Reformats labels based on the settings in this Dragged State Components category.
When you select Stacked Text, all blocks, lines, ticks, and direction arrows are removed. The text components are stacked vertically in the order they were defined in the label style.

**NOTE** The Summary tab of the Label Style Composer dialog box lists the components in the order in which they were defined.

Border Visibility

Specifies whether the border is visible when a label is dragged from its original position.

Border Type

Specifies the shape of the border.

- **Rectangular**: Draws a rectangle around text.

- **Rounded Rectangular**: Draws a rectangle with round corners around text.
  The radius used to create the round corners of the rectangle is calculated by adding the gap value and
  half the overall text height (including descending, subscript, and superscript characters).

Border And Leader Gap

Specifies the distance between the leader and text.

Text Height

Specifies the plotted height for text.

Leader Attachment

Specifies the location where the leader hook is drawn relative to the label content.

Leader Justification

Specifies how label text is justified in relation to the leader.

- **True**: Text is left-justified when the leader is on the left of the text, and is right-justified when the
  leader is on the right.

- **False**: Text is left-justified regardless of the leader location.

**Related procedures:**

- Managing Dragged Properties for Label Styles (page 1517)


# Summary Tab (Label Style Composer Dialog Box)

Use this tab to review all settings for a label style.

Label Content Summaries (Component 1, and so on)

Lists the label components as defined in the label style. The components are listed numerically in the
order in which they were created. For example, if the label style contains multiple components, the
categories are Component 1, Component 2, Component 3, and so on.
The properties are listed under each component prefixed with the category of the component. The values
of each of the properties replicate those made in the General, Layout, and Dragged State tabs of the Label
Style Composer dialog box.

Collapse All Categories

Collapses the property categories to show only the top level item.

Expand All Categories

Expands categories to show all properties.

Override All Dependencies

Selects all the check boxes ☑ in the Override column, which prevents the style from being changed if
values change at a higher-level setting (either in label settings or in a parent label style).

**Status Update**

At the bottom of the Summary tab the status of overrides is displayed as you select an item. For more
information, see Managing Overrides in Label Style Dialog Boxes (page 1960).

# Managing Overrides in Label Style Dialog Boxes

Use override options in label style dialog boxes to disconnect a setting from higher-level settings, and to restore settings to the higher-level settings.

Property column

Contains label categories and label property names.
Category names (in bold text) contain a list of one or more label properties (in normal text). Click plus (+) or minus (-) next to a category name to expand or collapse the list of property names.

Value column

Displays the current value for each property. When you change the default value in this column, a check mark is displayed in the Override column.

---

**NOTE** Even if you don't want to change the default value, you can manually select the check box in the Override column to prevent the setting from being affected by changes to a higher-level setting.

---

Override column

Identifies whether a property is overridden at the current level. A clear check box ☐ indicates the value is the same as the higher level default setting.

A selected check box ☑ indicates that the value has been explicitly set at the current level. If you click the check box to clear it, the setting is restored to the default, higher-level setting.
You can clear the check box to clear the override and set the value back to the value of its parent setting. Even if you don't specifically change a value, you can select the check box manually to disconnect the setting from the higher-level setting. This prevents it from changing if the higher-level setting changes.

Child Override column

⬇ Indicates that the value setting has been overridden at a lower level.

⬇ Displays when you click ⬇ and removes lower-level overrides in other settings and styles when you click OK or Apply.

---

**WARNING** Using this option resets the property in subordinate settings and/or styles even if they are locked.

---

Lock

Prevents the property value from being changed at a subordinate level. To lock a value, click 🔓 to change it to 🔒 . The lock cannot be set if there is a child override set.

⬇ Override All Dependencies

Selects all the check boxes ☑ in the Override column, which prevents the properties from being affected by changes to higher-level settings.

**Status Update**

At the bottom of the Summary tab in the Label Style Composer dialog box and in the three different Edit Label Style Defaults dialog boxes, the override status is displayed as you select an item.

Property

Displays the selected property name followed by a single line description of the property.

Parent

Displays the name of the parent from which the default value was derived. This is always the next-highest level at which the value was explicitly specified or overridden. Parents can include the following:

- Drawing Settings

- <Feature Name> Settings

- <Label Style Type Name> Settings

- <Parent Label Style Name>

For example, if you are reviewing a label style, the parent may be the Drawing Settings if none of the intermediate settings were changed. However, if an intermediate setting was changed, for example, at the feature level, then the <Feature Name> settings is listed as the parent.

Parent Value

Displays the value of the property as defined in the parent setting or style.

**Related procedures:**

- The Hierarchy of Label Settings (page 1468)

# Text Component Editor Dialog Box

Use this dialog box to add content to a label style.

You can override the content of an individual label by adding property fields for dynamic text, entering static text, and specifying formatting options. These edits do not modify the style being used by the label.

Access this dialog box on the Layout tab of the Label Style Composer by clicking the button in the Contents Value column, as shown in the following illustration:



**See also:**

- Text (page 1502)

- Adding Text Components to Labels (page 1503)

- Formatting Text (page 1505)

- Adding Property Fields to Label Text Components (page 1507)

- Editing Text in the Text Component Editor (page 1507)

## Common Elements of the Text Component Editor Dialog Box

Some common elements in the Text Component Editor are available when either the Properties or Format tab is selected.

Text Component Editor Window

Contains the property fields for the text component.

When you select a property and click  , the property field is inserted into the window. For more information, see Properties Tab (Text Component Editor Dialog Box) (page 1962).

You can also type text directly into this window. For more information, see Text Editor (Text Component Editor Dialog Box) (page 1964).

Import Text

Imports static text into the Text Component Editor window. Click Import Text to open the Select File dialog box. You must select a file that is either ASCII or RTF format.
You can format imported text by using the options on the Format tab (page 1963).

## Properties Tab (Text Component Editor Dialog Box)

Use this tab to add property fields to text components.

Property fields control the label content. Labels that contain property fields are linked to drawing objects and update dynamically when the drawing changes. For more information, see Property Fields (page 1507).

When a property field is added to a text component in the text editor window, the text content is constructed of modifier codes within a two character bracket symbol <[########]>.

**WARNING** The angle and square bracket combination is reserved for use by property fields. If you enter an angle and square bracket combination manually, it will cause an error in the text component. Also, manually editing modifier codes causes an error in the property field. The errors are displayed as three question marks (???) in the label when the property fields are processed. To change a property field in the Text Component Editor window, delete the property field completely from the window (including the angle and square bracket combinations), and insert a new property field.

Properties

Specifies a property to insert into the label text component. Properties are based surface spot elevation. When you select a property, Modifiers and their Values are displayed in the grid below it.

**NOTE** Edit the values as needed before clicking the Add Properties button. If you make changes to the values after you insert the property, you must delete and reinsert the property.

Filter

Opens the Property Filter (page 1963) dialog box where you can select which type of properties to include in the Properties list. This restricts the Properties list so it shows only relevant properties for the label type you are creating. For example, this is useful when you are creating a pipe label and want to restrict the properties shown to those relevant only to a particular type, such as circular pipes. Not available for all label types.

Add Properties

Inserts the property field into the Text Component Editor window at the cursor location.

Modifier

Lists all modifiers supported by the selected property field. For example, these modifiers may include unit, precision, and angle format. For more information, see Property Field Modifiers (page 1509).

Value

Lists the value for each modifier. The values and the method of formatting them vary according to the type of modifier.

**Related procedures:**

■ Text (page 1502)

■ Adding Text Components to Labels (page 1503)

## Property Filter Dialog Box

Use this dialog box to restrict the available properties in the Properties list on the Layout tab of the Text Component Editor dialog box.

By selecting filter options in this dialog box, you can limit the contents of the Properties list so you can more easily see the properties that are relevant to the label type you are composing.

Property

Lists the categories of filters available.

Value

Specifies the filter to use. Select a specific filter or specify one of the following:

■ **<All>**: Resets the filter to include all properties. A restrictive filter remains active until you use this option.

■ **<None>**: Excludes all such properties from the Properties list.

**NOTE** After you select a filter, it remains active for new label styles also. If you do not see the properties you expect in the Text Component Editor, click the Filter icon to determine whether the Properties list is filtered. Use the <All> option to turn filters off.

## Format Tab (Text Component Editor Dialog Box)

Use this tab to define how label text is formatted.

Text Style

Displays the text style on which all text components are based.

**NOTE** The text style for a label is specified on the General (page 1942) tab of the Label Style Composer dialog box.

Justification

Specifies how the entire text component is justified.

■  **Left**: Text is left justified.

■  **Center**: Text is centered.

■  **Right:** Text is right justified.

Font

Specifies the font for new or selected text.

TrueType® fonts  are listed by font family name. AutoCAD compiled shape (SHX) fonts  are listed by the name of the file in which the fonts are stored. Custom fonts or third-party fonts are substituted with Autodesk-supplied proxy fonts. A custom SHX font is available for character formatting overrides only if it is defined for a text style in the drawing.

Color

Specifies the text color.

Select a color from the list or click  to open the Select Color dialog box.

### Formatting options

**B** Bold

Turns bold formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

*I* Italic

Turns italic formatting on and off for new or selected text. This option is available only for characters using TrueType fonts.

U Underline

Turns underlining on and off for new or selected text.

◁ Undo

Reverses the most recent action in the Text Component Editor, including changes to either text content or text formatting.

**NOTE** You can also press Ctrl+Z to undo the most recent change.

▷ Redo

Reverses the effects of the last Undo that you performed, including changes to either text content or text formatting. The Redo option must immediately follow the Undo command.

ᵇ/ₐ Stack/Unstack

Stacks selected text that is formatted using stack characters, or unstacks stacked text. For more information, see Creating Stacked Characters (page 1505).

[ Symbol ▼ ]

Inserts special characters and symbols.

**NOTE** Diameter symbols appear as %%c in the Text Component Editor window but are converted to Ø in labels.

The Other option in the Symbol list displays the Character Map dialog box. To insert a character from this dialog box, select it, and then click Select. Select all the characters you want to use and then click Copy. Then right-click in the Text Component Editor window, and click Paste.

**Related procedures:**

- Formatting Text (page 1505)
- Creating Stacked Characters (page 1505)
- Editing Text in the Text Component Editor (page 1507)


## Text Editor (Text Component Editor Dialog Box)

Use the Text Editor window, which is the area on the right side of the Text Component Editor dialog box, to construct a text component.

The Text Editor window has three purposes:

- It contains the property fields that you specify for the text component
- It allows you to add static text to a label

■ It allows you to select text for formatting and editing

Property fields should not be edited or the label will not be created correctly. You can, however, enter text in the window and use the following shortcut menu items to format it:

Shortcut Menu

Select the text, and right-click to display a shortcut menu for quick formatting options. The menu is context sensitive to cursor position, text selection, and stacked text.
The shortcut menu is divided into three parts. The top part has standard editing commands: Undo, Redo, Cut, Copy, and Paste.
The lower part of the shortcut menu contains specific formatting options.

■ **Select All**: Selects all text in the editor window.

■ **UPPERCASE**: Changes all selected text to uppercase characters.

■ **lowercase**: Changes all selected text to lowercase characters.

■ **Remove Formatting**: Removes any formatting, such as underline, bold, italic, and word-wrapping.

■ **Combine Paragraphs**: Removes all line returns.

■ **Stack**: Stacks selected text. This option appears only when text that is formatted with stacking characters is selected. For more information, see Creating Stacked Characters (page 1505).

■ **Unstack**: Unstacks text. This option appears only when stacked text is selected.

■ **Properties**: Displays the AutoCAD Stack Properties dialog box. This option appears only when stacked text is selected.

**Related procedures:**

■ Editing Text in the Text Component Editor (page 1507)

# Label Style Control

Use this control to create a new label style, edit the current selection, or select a style from the drawing.

The Label Style control is available in many dialog boxes.

Create New

Creates a new label of the same type in the Label Style Composer dialog box.

Copy Current Selection

Creates a copy of the current label style.

Create Child Of Current Selection

Creates a child from the current label style.

Edit Current Selection

Opens the Label Style Composer, where you can edit the current label style.

Pick From Drawing

Displays a pickbox so you can select a label style directly from the drawing.

# Label Properties Dialog Box

Use this dialog box to change the properties of labels in the drawing.

Different properties are available, depending on the type of label selected. The most general properties are listed first.

**General**

Reset Label Defaults

Returns all properties to original values. Use this option to reset the properties after you have made changes. This option works only within the current instance of the command; it does not restore previously-changed properties.

Label Style

Displays the style used by the selected label. Click the Value column, and then click  to open the Label Style dialog box and to select a different style.

Visibility

Specifies whether the label is visible in the drawing.

■ **True**: Displays the label.

■ **False**: Hides the label. This setting deletes the label unless it is reset before you close the dialog box.

Flip Label

Specifies whether the label is flipped.

■ **True**: Flips the label components to the opposite side of a line or curve object.

■ **False**: Leaves label as is.

Label Is Pinned

Specifies whether the label is fixed to its location.

■ **True**: Pins a label to its current location regardless of edits to the object referenced by that label.

■ **False**: Label can be moved if the object is edited.

**Bearing Labels**

Reverse Direction

Reverses the direction of labels that contain bearings.

■ **True**: Reverses direction label.

■ **False**: Leaves direction label as is.

**Referenced Text Objects**

Referenced Text Objects

Specifies the object to refer to in labels that contain Referenced Text. Click in the Value column and then click  to display the Property Field Object dialog box. Select the object to refer to and click OK.

**Surface Spot Elevation Labels**

**Profile View, Section View, Surface Spot Elevation Labels**

Point Style

   Specifies a point style which controls the appearance of the label insertion point for profile view, section view, and surface spot elevation labels. Select a point style from the list.

## Alignment Offset Labels

Leader Attachment Option

   Specifies the point attachment location for the label leader line.

   - **Point**: Leader line attaches to the point.

   - **Marker Extents**: Leader line attaches to the extents of the marker.

Marker Style

   Displays the style used by the currently selected marker. Click in the Value column and then click ⊡ to display the Marker Style dialog box and to select a different style.

Fixed XY

   Specifies the label position for fixed point alignment station/offset labels.

   - **True**: Moves the label with the alignment.

   - **False**: Leaves label as is.

## Profile View and Section View Labels

Profile 1 Object

   Specifies a profile to be used as "Profile 1" in the profile view labels. Use this option to include specific profile data in a label. For more information, see Setting Up Profile and Section View Labels (page 1526).

Profile 2 Object

   Specifies a profile to be used as "Profile 2" in the profile view labels.

Section 1

   Specifies a section to be used as "Section 1" in the section view labels.

Section 2

   Specifies a section to be used as "Section 2" in the section view labels.

Dimension Anchor Option

   Specifies the location of the dimension line anchor used to position dimension lines for certain profile view labels, such as vertical curve labels.

   - **Default**: Leaves the anchor position as is.

   - **Above**: Moves the dimension line anchor up the distance specified in the Dimension Anchor Value.

   - **Below**: Moves the dimension line anchor down the distance specified in the Dimension Anchor Value.

   - **Fixed**: Places the dimension line anchor at a fixed elevation on a profile view. Specify the elevation in the Dimension Anchor Value field.

Dimension Anchor Value

   Specifies the dimension line anchor value to be applied using the Dimension Anchor Option.

**Pipe and Structure Labels**

Alignment Name

Specifies the alignment that the pipe or structure references for labeling purposes. All alignments in the drawing are listed, as well as <Default>. <Default> is the alignment that the part was associated with when it was created. Changing the alignment name in the label properties changes only the data that appears in labels; it does not change the actual association between the part and alignment.

**Related procedures:**

■ Modifying Labels in a Drawing (page 1533)

# Label Set Dialog Box

Use this dialog box to create a group of label styles for alignments, profiles, or sections.

# Information Tab (Label Set Dialog Box)

Use this tab to change the label set name and description information, and to review details, such as when the label set was most recently modified.

Name

Specifies the name of the current label set.

Description

Specifies the description of the current label set.

Created By

Displays the AutoCAD login name of the person who created the label set.

Date Created

Displays the date and time the label set was created.

Last Modified By

Displays the AutoCAD login name of the person who last modified the label set.

Date Modified

Displays the date and time the label set was last modified.

**Related procedures:**

■ Creating a Label Set (page 1547)

# Labels Tab (Label Set Dialog Box)

Use this tab to select the label styles to add to an alignment, profile, or section label set.

After you add a label style to a set, you can define certain properties for the label style, such as increment. The properties vary depending on whether you are defining an alignment, profile, or section label set.

Type

Specifies the type of label to add to the label set.

<Type> Label Style

Displays the label styles available for the specified label type. Select a label style from this list, and click Add to add it to the label set.

Label Style Control

Creates a new label style, edits the current selection, or selects a style from the drawing.

 Style Detail

Opens the Style Detail dialog box where you can preview the selected label style.

 Add Label Style

Adds the label style selected in the Label Style list to the label set.

 Delete Label Style

Removes a label style from the label set. Select the label style to delete by clicking in the row in the lower part of the dialog box.

## List of Selected Styles

When you add a label style to a set, it is inserted into the list in the lower part of the dialog box. You can use this area to define properties for the label style and to change the selected label style.

Type Column

Displays the label types in the current label set.

Style Column

Specifies the label style to include in the set. Click  to open the Pick Label Style dialog box.

Increment Column

Specifies the increment at which to insert major and minor station labels.

> **NOTE** This setting is available for station labels only.

Geometry Points To Label Column

Specifies the geometry points to label for the Geometry Points, Profile Geometry Points, and Superelevation Critical Points label types. You can label a selection of geometry points in a label type using a selected style. You can then add another instance of the same label type, which can label a second selection of geometry points using a different style. Click  to select the geometry points to label.

Stagger Labels Column

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left. The staggering options are available for the following labels: Profile (Grade Breaks, Major/Minor Station, Horizontal Geometry), and Section labels (Major/Minor Offsets and Grade Break).

Stagger Line 1 Height Column

Specifies the height of the first profile or section view data band label stagger line.

Stagger Line 2 Height Column

Specifies the height of the second profile or section view data band label stagger line.

## Profile Label Properties

After you add a profile label style to a label set, you can specify the following property values that are unique to profile label sets:

Dim Anchor Opt

Specifies the location of the dimension line anchor used to position dimension lines for certain profile view labels, such as vertical curve labels.

- **Distance Above**: Moves the dimension line anchor up the distance specified in the Dim Anchor Val field.

- **Fixed Elevation**: Places the dimension line anchor at a fixed elevation on a profile view. Specify the elevation in the Dim Anchor Val field.

- **Distance Below**: Moves the dimension line anchor down the distance specified in the Dim Anchor Val field.

Dim Anchor Val

Specifies the dimension line anchor value to be applied using the Dim Anchor Opt.

Weeding

Specifies a label exclusion distance for grade break labels only. Use this setting to remove overlapping labels and make it easier to read the remaining labels. If another break occurs within the specified distance from a given grade break, it is not labeled.

**NOTE** This setting is available for Grade Break labels only.

**Related procedures:**

- Creating a Label Set (page 1547)

# Add Labels Dialog Box

Use this dialog box to label objects in a drawing.

The Add Labels dialog box is common to most objects. Options available in the dialog box change depending on the feature selected.

Feature

Specifies the feature to label.

Label Type

Specifies the type of label to add. This list changes depending on the feature selected.

Edit Command Settings

Opens the Edit Command Settings dialog box where you can specify the default styles to use for each type of label.

Label Style list

Specifies the label style to use for new labels.

Label Style Control

Creates a new label style, edits the current selection, or selects a style from the drawing.

Style Detail

Displays the Style Detail dialog box where you can preview the selected label style.

Marker Style list

Specifies a marker style which controls the appearance of the label insertion point for profile view, section view, and surface spot elevation labels. Select a marker style to use from the list.

Reference Text Object Prompt Method

Specifies the method for selecting Referenced Text (page 1509) objects. Used only for label types that support Referenced Text.

- **Command Line**: When inserting a label that contains Referenced Text, you are prompted at the command line to select the objects to refer to in the label.

- **Dialog**: When inserting a label that contains Referenced Text, the Label Properties dialog box is displayed. Click in the Value column and then click ⊡ to display the Property Field Object dialog box. Select the object to refer to and click OK.

- **Suppress Prompts**: When inserting a label that contains Referenced Text, all prompting for objects is suppressed. No objects are specified for Referenced Text fields that may exist in the label style. You can edit the label properties later to specify the objects to refer to.

Table Tag Numbering

Accesses the Table Tag Numbering dialog box.

Add

Inserts labels of the selected style into the drawing after you select the object. The prompts that are displayed after you click Add vary based on the label style.

**Related procedures:**

- Adding Labels to Drawings (page 1522)

# Table Tag Numbering Dialog Box

Use this dialog box to define how line, curve and spiral components are numbered and renumbered.

There are two numbering counters:

If you create a number of components at the same time, the Starting Number property is used to generate the starting number and the numbering automatically increments by one unless you edit the increment number. The starting number is only changed automatically when a table tag is created and numbered. If there is a duplicate with the starting number, it increments up by the increment amount. If there is still a duplicate, it increments up, until an unused tag number is found and applied. The starting number is then set to the next number according to the increment (even if it is a duplicate).

The starting point for the increment is always the starting number. For example, if the Starting Number is 11, and the Increment is 10 and you attempt to create a tag but the starting number 11 is already used. The next starting number is now 21. If that is also used, it will increment to 31. If 31 is available, the tag will be numbered with 31, and the new starting number set to 41.

---

NOTE When creating tags, the renumbering starting number and increment will not be changed (unless you manually change them in the Table Tag Numbering dialog box.

---

### Table Tag Creation

Lines Starting Number

Specifies the starting number for new lines to be created in the site. The numbering is incremented with creation of new lines.

Lines Increment

Specifies the difference between numbers in the numbering sequence.

Curves Starting Number

Specifies the starting number for new curve tags to be created in the site. The numbering is incremented with creation of new tags.

Curves Increment

Specifies the difference between numbers in the numbering sequence.

Spirals Starting Number
  Specifies the starting number for new spiral tags to be created in the site. The numbering is incremented with creation of new tags.

Spirals Increment
  Specifies the difference between numbers in the numbering sequence.

**Table Tag Renumbering**

Lines Starting Number
  Specifies the starting number for new line tags to be renumbered.

Lines Increment
  Specifies the difference between numbers in the numbering sequence..

Curves Starting Number
  Specifies the starting number for new curve tags to be renumbered.

Curves Increment
  Specifies the difference between numbers in the numbering sequence.

Spirals Starting Number
  Specifies the starting number for new spiral tags to be created on alignments in the site. The numbering is incremented with creation of new tags.

Spirals Increment
  Specifies the difference between numbers in the numbering sequence.

**Related procedures:**

■ Creating a Site (page 723)

■ Editing Site Properties (page 724)


# Select Text Component Dialog Box

Use this dialog box to select a label text component to override.

Text Component
  Lists the text components in the label style. Select the component to override and click OK.
  A check mark appears next to any component that has already been overridden.

**Related procedures:**

■ Overriding Label Text (page 1535)


# Select Overridden Text Component Dialog Box

Use this dialog box to select an overridden text component in order to clear the overrides.

Text Component
  Lists the overridden text components in the label style. Select one or more components and click OK.
  You can use the Shift and Ctrl keys to select multiple components.

**Related procedures:**

■ Overriding Label Text (page 1535)

# Expressions Dialog Box

Use this dialog box to set up mathematical expressions that you can insert into label styles.

For example, you can set up an expression for parcel segments that converts feet to meters:

```
{Segment Length} *.3084
```

For other examples, see Expression Examples (page 1549). For procedures, see Setting Up Expressions (page 1548).

**Tips for creating expressions**

■ Do not enter functions, constants, or logical operators inside the curly brackets that surround Properties such as {Segment Length}.

■ Do not enter plain text, such as "meters" in expressions. You can add this text later when you set up a label style that includes the expression.

To set up an expression, you can use the buttons in the dialog box, and the keypad on your keyboard. There are menus in the dialog box from which you can choose Properties and functions to insert into the expression.

If you enter a combination of items that is invalid, you will receive a parse error when you click OK.

Name

Specifies a name for the expression. It is recommended that you use a clear naming convention such as "Parcel line expression: Length in meters". Expressions are listed by name along with all the other label properties in the Label Style Text Component Editor.

Description

Specifies a description that appears in the Settings tab item list view.

Editor Box

Displays the expression as defined from the functions and properties. You can also click in this box and type characters such as numbers.

Calculator Buttons

Enter numbers and operators (page 1848) into the Editor box to define the expression.

Insert Property

Inserts a property into the expression. Click the icon to display a list of properties relevant to the expression type. Select a property to insert it into the expression. These are the same Properties (page 1507) that are available for label styles.
Properties are inserted inside brackets. Do not add operators or functions within these brackets.

Insert Function

Inserts a function into the expression. See Functions (page 1846) below for more information.

Format Result As

Specifies how the expression results are formatted.

■ **Double**: This value is formatted using the Unitless type.

> **NOTE** The following are all of type Double but are formatted according to the settings on the Ambient Settings tab (page 1856) in the Drawing settings:

■ Angle

■ Area

■ Coordinate

- Dimension

- Direction

- Distance

- Elevation

- Grade/Slope

- Latitude

- Longitude

- Station

- Volume

## Functions

NOTE Functions can be either upper case or lower case but not mixed case. The editor buttons will always insert functions in upper case. Unless otherwise stated, theta is in radians.

| | |
|---|---|
| ABS(x) | Returns absolute value of x |
| ACOS(x) | Returns arccosine of x |
| ASIN(x) | Returns arcsin of x |
| ATAN(x) | Returns arctangent of x |
| ATAN2(y,x) | Returns arctangent of y/x in the correct quadrant based on signs of x and y |
| CEIL(x) | Returns the smallest integer that is greater-than or equal to x |
| COS(theta) | Returns the cosine of theta |
| COSH(theta) | Returns the hyperbolic cosine of theta |
| COT(theta) | Retruns the cotangent of theta |
| COTH(theta) | Returns the hyperbolic cotangent of theta |
| CSC(theta) | Returns the cosecant of x |
| CSCH(theta) | Returns the hyperbolic cosecant of x |
| DEG2GRD(theta) | Converts theta in degrees to gradients |
| DEG2RAD(theta) | Converts theta in degrees to radians |
| DRVSTN(x) | Returns x at the derived station along the corridor |
| EXP(x) | Returns exponential of x |

| FLOOR(x) | Returns the largest integer that is less-than or equal to x |
| --- | --- |
| FMOD(x,y) | Returns the floating point remainder of x/y |
| GRD2DEG(theta) | Converts theta in gradients to degrees |
| GRD2RAD(theta) | Converts theta in gradients to radians |
| IF(test,true_val,false_val) | Evaluates test - if test is non-zero evaluates and returns true_val else evaluates and returns false_val. True_val and false_val can be any expression.<br>For example, where x is a Property:<br>IF(x=0,1,sin(x)/x)<br>The above tests x to see if it is zero, and if it is, the expression returns 1. If x is non-zero, the expression returns sin(x)/x. |
| LOG(x) | Returns the log (base e) of x |
| LOG10(x) | Returns the log (base 10) of x |
| MAX(a,b) | Returns maximum value of a and b |
| MIN(a,b) | Returns minimum value of a and b |
| POW(x,y) | Returns x raised to the y power |
| POW10(x) | Returns x raised to 10 |
| RAD2DEG(theta) | Converts theta in radians to degrees |
| RAD2GRD(theta) | Converts theta in radians to gradients |
| ROUND(x) | Rounds x to the nearest inetger |
| SIN(theta) | Returns the sin of theta |
| SEC(theta) | Returns the secant of theta |
| SECH(theta) | Returns the hyperbolic secant of theta |
| SINH(theta) | Returns the hyperbolic sin of x |
| SQR(x) | Returns x squared (i.e. x*x) |
| SQRT(x) | Returns the square root of x |
| TAN(theta) | Returns the tangent of theta |
| TANH(theta) | Returns the hyperbolic tangent of theta |
| TRUNC(x) | Truncates x to an integer value |

## Logical Operators

**NOTE** Logical operators return 1.0 for true and 0.0 for false. Since they return values, you can perform logical AND and OR with addition and multiplication. For example: ((a<b) + (b>c)) tests for a < b or b > c. ((a<b)*(b>c)) tests for a<b and b>c.

| Symbol | Operator Definition |
|--------|---------------------|
| = | logical equals |
| != | logical not equals |
| ! | logical not |
| < | logical less-than |
| > | logical greater-than |
| <= | logical less-than or equals |
| >= | logical greater-than or equals |
| And | & Boolean "and" |
| ~ | logical approximately equal |
| <~ | logical less-than or approximately equal |
| >~ | logical greater-than or approximately equal |
| Or | \| - Boolean "or" |
| + | binary addition |
| - | binary subtraction |
| * | binary multiplication |
| / | binary division |
| - | unary minus |
| + | unary plus |
| ^ | power |

**Related procedures:**

■ Expressions (page 1547)

# LandXML Dialog Boxes

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Use the following links to access information about the AutoCAD Civil LandXML dialog boxes.

## Import LandXML Dialog Box

Use this dialog box to select the LandXML data to import into AutoCAD Civil.

Alignments Site

Specifies a site for imported alignments. Select a site from the drop-down list or click ![icon] to select an object (parcel, alignment, or grading) contained in the site to which you want to add the new objects.

NOTE The default selection is <None>, which places the alignments in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

Parcels Site

Specifies a site for any imported parcels. Select a site from the drop-down list or click ![icon] to select an object (parcel, alignment, or grading) contained in the site to which you want to add the new objects.

NOTE If no sites exist in the drawing, the objects are placed in a default site.

Edit LandXML Settings

Opens the LandXML Settings dialog box (page 1978).

Data Tree

Lists the LandXML header information for each data component that can be imported. Expand each of the collections to display all of the subcomponents.
Select or clear the check boxes to the left of each data component name to filter the data that is imported into AutoCAD Civil. By default, all data components are selected for import.

NOTE The check boxes have a tri-state display. If only some items are selected under a collection, the check box is shaded ![icon] . If all items are selected, the check box is selected ![icon] . If all items below the collection are cleared, the check box for the collection is cleared ![icon] .

OK

Converts LandXML data into AutoCAD Civil objects. Imports each object from the LandXML file into the AutoCAD Civil drawing using the name as specified in the LandXML file.

Each object uses the appropriate creation behavior based on the command setting for the object being imported. For example, if the Surface Default Style is set to High Altitude, any surfaces imported with this operation will use the High Altitude style.

**NOTE** As each object is imported, the Event Viewer Vista (page 2455) dialog box can display the status of each operation. For more information, see The Event Viewer Vista (page 1626).

Browse

Opens the Open dialog box. Select another LandXML file from which to import data.

**Related procedures:**

■ Importing LandXML Drawing Data (page 1681)

# LandXML Settings Dialog Box

Use this dialog box to manage a variety of settings related to the LandXML import and export properties.

## Import Tab (LandXML Settings Dialog Box)

Use this tab to define how data is imported from a LandXML file to AutoCAD Civil.

You can set the data translation, rotation, and conversion settings. In addition, there are options to control the import of point, surface, and pipe network data.

### Translation

Translate

Specifies whether to translate data to new coordinates when importing the data from the LandXML file. Select On to translate the data on import. For more information, see Translating and Rotating LandXML Data (page 1679).

**NOTE** If Translate is set to Off, all other data translation options are unavailable.

Base Point Northing

Specifies the base point northing coordinate from which all import values are translated.

**NOTE** The units used for base point and translated values are set at the drawing level in the Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856).

Base Point Easting

Specifies the base point easting coordinate from which all import values are translated.

Base Point Elevation

Specifies the base point elevation from which all import values are translated.

Translated Coordinate Northing

Specifies the northing coordinate value used to translate data from the base point.

Translated Coordinate Easting

Specifies the easting coordinate value used to translate data from the base point.

Translated Coordinate Elevation

Specifies the elevation value used to translate data from the base point.

**Rotation**

Rotate

Specifies whether to rotate data when importing it from the LandXML file. Select On to rotate the data on import.

NOTE If Rotate is set to Off, all other data rotation options are disabled.

Angle

Specifies the angle about the base point where the LandXML data is rotated.

NOTE The angle units are set at the drawing level in the Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856).

Direction

Specifies the direction of rotation. Select this option to calculate the angle in a clockwise or counter-clockwise rotation about the base point.

## Point Import Settings

Point Description

Specifies how to map the 'desc' and 'code' attributes to the point descriptions:

- **Use "Code" Value**: Uses the "code" attribute value for the point description.

- **Use "Desc" Value**: Uses the "desc" attribute value for the point description.

- **Use "Code" Then "Desc"**: Attempts to use the "code" attribute value for the point description. If a point does not have a "code" attribute, the "desc" attribute value, if available, is used.

- **Use "Desc" Then "Code"**: Attempts to use the "desc" attribute value for the point description. If a point does not have a "desc" attribute, the "code" attribute value, if available, is used.

## Surface Import Settings

Surface Data

Specifies how to import the data:

- **Quick Import**: Imports the surface information as points. Creates a surface that has identical northing/easting/elevation points as the surface definition in the LandXML file. However, the triangulation may be different from the original surface.

- **Full Import**: Imports both the points and the surface faces, if available. Creates the surface information from the face edges and creates a surface that is identical to the surface definition in the LandXML file.

  NOTE If the LandXML surface you intend to import has only breaklines (for example, no other points and faces definition data is present), the resulting AutoCAD Civil surface is built from the breakline data. However, the breaklines are not available for editing (that is, they do not appear in the Prospector tree).

Create Snapshot After Import

After importing surface data, a snapshot can automatically be created. By creating a snapshot after importing a LandXML file, AutoCAD Civil does not attempt to find and open the LandXML file each time the surface is built. Select On to create a snapshot. For more information on snapshots, see Using Surface Snapshots (page 681).

Create Source Data In Drawing

Imports the surface source data (breaklines, contours, points, and chains) into the drawing as 3D polylines and points.

- Breaklines, contours, and chains are created as 3D polylines.

■ DataPoints are created as COGO points in a point group named <surface name>_DataPoints.

This data is created on the following layers prefixed with the surface name:

■ <surface name>_Breaklines

■ <surface name>_Contours

■ <surface name>_Chains

■ <surface name>_DataPoints_PntList3D

---

**NOTE** This data is not added to the surface definition in the Prospector tree. You can manually add it if desired, or create a different surface from it for comparison purposes.

---

### Pipe Network Import Settings

Parts List For Part Family Size Matching

Specifies the pipe network parts list that is used during import to match pipe network part types in a LandXML file with pipe network part types in a AutoCAD Civil parts list. Review these settings prior to import to verify that settings are appropriate for your data. For more information, see Part Catalog and Parts Lists (page 1237).

Match <Part Type> With:

Specify individual part types to use when importing pipe network parts from a LandXML file into AutoCAD Civil. Review these settings prior to import to verify that settings are appropriate for your data. You may use the defaults, or click ⊡ to specify other parts in the Part Catalog dialog box.

### Conflict Resolution Settings

Conflict Resolution

Specifies how objects with the same name are handled.

■ **Skip**: Does not import conflicting objects.

■ **Update**: On import, replaces unlocked drawing object with object in LandXML file.

■ **Append**: Imports conflicting objects but renames the imported copy using the default name format that is specified in the feature settings.

### Default Diameter Units

On import, AutoCAD Civil reads the <LandXML>.<Units>.diameterUnits, and if they are defined, those units are used for both pipes and structures. If units are not defined, the following unit settings are applied. Either the Imperial or Metric diameter settings are applied, based on the defined <LandXML>.<Units>.linearUnit in the file.

Imperial

Defines the default imperial pipe and structure diameter units that are applied when units are not specified in the source LandXML file.

Metric

Defines the default metric pipe and structure diameter units that are applied when units are not specified in the source LandXML file.

**Related procedures:**

■ Importing LandXML Drawing Data (page 1681)

■ Viewing and Editing LandXML Drawing Settings (page 1678)

# Export Tab (LandXML Settings Dialog Box)

Use this tab to define how data is exported from AutoCAD Civil to a LandXML file.

## Identification

Write Identification Elements

Specifies whether the identification properties are written to the LandXML file. Select On to write populated identification properties to the LandXML file.

**NOTE** If Write Identification Elements is set to Off, all other identification options are disabled.

Creator

Specifies the creator of the LandXML file.

Email Address

Specifies the creator's email address.

Company

Specifies the creator's company.

Company URL

Specifies the company's URL.

## Data Settings

Imperial Units

Specifies how to tag LandXML data when working in Imperial units:

- **International Foot**: Tags the data in the LandXML file as International feet.

- **Survey Foot**: Tags the data in the LandXML file as Survey feet.

Angle/Direction Format

Specifies the units of the exported angles and directions:

- **Degrees Decimal (DDD.DDDDDD)**: Exports angles and directions as decimal degrees. Decimal degrees express the minutes and seconds of an angle as its decimal equivalent. For example, 3°30'36" equals 3.51 decimal degrees.

- **Degrees Decimal dms (DDD.MMSSSS)**: Exports angles and directions using the degrees, minutes, and seconds convention, in which the numbers after the decimal are read as minutes and seconds, not decimal degrees. For example, 3°30'36" equals 3.303600.

- **Grads**: Exports angles and directions using grads (where one grad equals 1/100 of a 90° angle, or 360° = 400 grads).

- **Radians**: Exports angles and directions using radians (where 2PI radians equals 360°).

**NOTE** Directions are always reported in azimuths. The LandXML schema does not support bearings.

Create Read-Only File

Specifies whether the LandXML file is created as a read-only file. This is done in two places: a tag is inserted into the LandXML, and the operating system file attributes on the file are set to read only. Select On to specify read only.

## Point Export Settings

Code Attribute

Specifies the field that is used for the 'code' attribute in the LandXML file:

- **Disabled**: Does not export the point description to a 'code' attribute.

- **Raw Description**: Exports a point raw description to the 'code' attribute.

- **Full Description**: Exports a point full description to the 'code' attribute.

---

**NOTE** Typically you would export the raw description to the 'code' attribute. For more information, see Point Description and Code Matching (page 1680).

---

Desc Attribute

Specifies the field that is used for the 'desc' attribute in the LandXML file:

- **Disabled**: Does not export the point description to a 'desc' attribute.

- **Raw Description**: Exports a point raw description to the 'desc' attribute.

- **Full Description**: Exports a point full description to the 'desc' attribute.

---

**NOTE** Typically, you would export the full description to the 'desc' attribute. If you select Disabled for both the Code Attribute and Desc Attribute, then no descriptions are exported. For more information, see Point Description and Code Matching (page 1680).

---

Skip Full When Same As Raw

Specifies whether to export the full description if it is the same as the raw description:

- **On**: Does not export the full description if it is the same as the raw description. For example, if the raw and full descriptions are the same, then only one value is exported. This would occur when points do not have defined description keys.

- **Off**: Exports both values.

Export Point References

Specifies whether to export point references. Point references substitute a known point name instead of using northing and easting coordinates. Select On to export point references.
For example, if the start point of a line element dose not have a point reference, it is written as follows:
<Start>5447.73530 4525.60643</Start>
If it has a point reference, it is written as follows:
<Start pntRef="371"></Start>
For more information, see Point Reference Export Options (page 1680).

Point Reference Tolerance

Specifies the tolerance value for point references.
For example, if you enter 0.001, only those points that are located within 0.001 units of the point of interest (for example, a start point) are exported as references.

Export Description Keys

Specifies whether the description key definitions are exported. Select On to export all description key definitions in the drawing. If there are description keys in the drawing, a message dialog box is displayed.

## Surface Export Settings

Surface Data

Specifies how to export the surface data:

- **Points Only**: Exports surface information as points. Creates a surface that has northing, easting, and elevation points that are identical to the surface definition. However, the triangulation may be different from the original surface.

- **Points and Faces**: Exports both the points and the surface faces, if available. Creates the surface information from the face edges and creates a surface that is identical to the surface definition.

Watersheds

Specifies whether to export watershed information with the surface data. Select On to export watershed information to the LandXML file.

**Parcel Export Settings**

Parcel Direction

Specifies the direction from the parcel start point for exporting parcel segments. Parcel segments are exported in sequence in either a clockwise or counter-clockwise direction.

- **Clockwise**: Exports parcel segments in a clockwise direction.

- **Counter-clockwise**: Exports parcel segments in a counter-clockwise direction.

**Alignment Export Settings**

Export Cross Sections

When set to On, exports cross section data if sample lines are defined for the alignment.

**Related procedures:**

- Exporting LandXML Drawing Data (page 1682)

- Viewing and Editing LandXML Drawing Settings (page 1678)

# Export To LandXML Dialog Box

Use this dialog box to select the data in the drawing to export in LandXML format.

This dialog box is used for exporting LandXML data as well as for generating LandXML-based reports.

Data Tree

Lists the AutoCAD Civil data that you can export in LandXML format. Expand each of the collections to display all of the subcomponents.
Select or clear the check box to the left of each feature name to filter the data that you want to export in LandXML format.

**NOTE** The check boxes have a tri-state display. If only some items are selected under a collection, the check box is shaded ☑ . If all the items are selected, the check box is selected ☑ . If all the items below the collection are cleared, the check box for the collection is cleared ☐ .

Specify LandXML Version

Specifies the LandXML schema version. Select from 1.0, 1.1, or 1.2.

Pick From Drawing

Specifies objects to export by selecting from the drawing.

**NOTE** If you access this dialog box by right-clicking a collection in the Prospector tree and selecting Export LandXML, the Pick From Drawing button is not available.

OK

Converts AutoCAD Civil objects to LandXML format and exports the LandXML data.

**Related procedures:**

- Exporting LandXML Drawing Data (page 1682)

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# Layers Dialog Boxes

# 57

Use the following links to access information about the dialog boxes for managing drawing layers.

## Object Layer Dialog Box

Use this dialog box to specify default layers for drawing objects. For example, when changing the layer for a point group in the Point Group Properties.

Base Layer Name

Displays the name of the layer to which the object is assigned in the Object Layers tab of the Drawing Settings dialog box. Use the layer selection button to select a different layer.

Layer selection button

Opens the standard Layer Selection dialog box. You can select a layer to change the base layer for the object. This button is disabled when the object layer settings (page 66) are locked.

Modifier

Specifies whether the layer name includes a text-string modifier, and if so, the location of the modifier:

- **None:** No modifier.

- **Prefix:** Modifier is added at the beginning of the layer name.

- **Suffix:** Modifier is added at the end of the layer name.

Modifier Value

Specifies the text string to use as the modifier. You can enter an asterisk (*) to include the object name as the modifier in the layer name and to put each named object on its own layer. For example, if C-ROAD is the name specified for alignment layers and you add -* as a suffix, an alignment named overpass is created on layer C-ROAD-OVERPASS.

Preview

Displays the final name of the layer for any newly created objects.

**Related procedures:**

- Layers (page 58)

# Layer Selection Dialog Box

Use this dialog box to select the layer on which to create objects of the selected type. Click anywhere in the row to select the layer, then click Apply.

When you select a layer from a source outside the current drawing, the layer is created in the current drawing. If you select a layer that exists in the current drawing, a warning dialog box is displayed, where you can overwrite the current layer definition or cancel the command.

Layer Source
   Specifies the drawing that contains the layer. Select one of the open drawings.

Layer
   Displays the name of the layer.

Color
   Displays the color of the layer.

Linetype
   Displays the linetype of the layer.

Lineweight
   Displays the lineweight of the layer.

Plot
   Displays whether the layer is plotted or not.

Plot Style
   Displays the plot style of the layer.

New
   Creates a layer using the Create Layer (page 1986) dialog box. This option is available only if the current open drawing is selected for the Layer Source.

# Create Layer Dialog Box

Use this dialog box to set the properties for the new layer.

**Layer Properties**

Layer Name
   Specifies the layer name.

Color
   Specifies the layer color.

Linetype
   Specifies the layer linetype. Select a linetype from the drawing or from those defined in the drawing.

Lineweight
   Specifies the layer lineweight.

Locked
   Specifies whether the layer is locked. On a locked layer, you cannot select or modify objects.

On
   Specifies whether the layer is visible.

Freeze
   Specifies whether the layer is frozen or thawed. A frozen layer is not updated.

Plot Style
   Specifies the plot style.
Plot
   Specifies whether the layer can be plotted.

# Lines and Curves Dialog Boxes

# 58

## Entity by Best Fit Dialog Box

Use this dialog box to create an AutoCAD, Alignment, or Profile entity by best fit. You can create the entity from a series of AutoCAD Civil points, AutoCAD points, existing entities, or by clicking on screen.

Use this dialog box to select the source objects from which you want to create a best fit entity. You may create the following types of entities by best fit:

### AutoCAD Entities

- Line

- Arc

- Parabola

### Alignment Entities

- Fixed Line

- Floating Line

- Fixed Curve

- Floating Curve

- Free Curve

### Profile Entities

- Fixed Tangent

- Floating Tangent

- Fixed Vertical Curve

- Floating Vertical Curve

■    Free Vertical Curve

---

**NOTE**  The From AutoCAD Points and From COGO Points input options are not available on the Parabola By Best Fit dialog box.

---

**Input Options**

From COGO Points

Specifies that two or more COGO points or survey points will be used for best fit entity creation.

From AutoCAD Points

Specifies that two or more AutoCAD points will be used for best fit entity creation.

From Entities

Specifies that one or more existing lines, arcs, polylines, feature lines, survey figures, or profile objects will be used for best fit entity creation.

By Clicking On The Screen

Specifies that the user will click points on screen for best fit entity creation.

Tessellate Arc Segments

Tessellates the selected sample arc segments.

Mid-Ordinate Tolerance

Specifies the mid-ordinate tolerance based on a distance across the tessellated arc segments. Enter a known value or click  to select two points along a curve.

**Related procedures:**

■    Creating Entities by Best Fit (page 582)

■    Adding Lines, Curves, and Spirals to an Alignment (page 898)

■    Adding Tangents and Curves to a Layout Profile (page 1059)

# Regression Data Vista (for Best Fit Entity Analysis)

Use this vista to view and edit the results of the best fit regression data.

**Entity Property Tree**

This tree lists attributes of the best fit entity. Attribute values cannot be edited.

**Regression Points Toolbar**

  Copy to Text Editor

Copies the regression data to the text editor that is associated with AutoCAD.

  Copy to Clipboard

Copies regression data to the clipboard with the appropriate formatting. This tool is useful for importing the regression data into a spreadsheet.

  Add More Points

Adds more regression points. This button launches the appropriate **Create Object by Best Fit** dialog box, which allows you to select more points to include in the regression analysis.

**✖** Delete Selected Points

Deletes selected points (rows) from the regression points table.

**↩** Undo

Reverses the last action.

**↪** Redo

Reverses the last undo command.

**▽** Out of Sync

Indicates that the best fit entity has been modified and its parameters do not match the best fit regression data. Click this button to force the entity to match the original best fit regression data.

**↻** Empty Current Regression Sample Data

Resets the regression points table and leaves the Regression Data dialog box open. New points may be

added by clicking **✛** .

**▣** Create Entity and Continue to Edit the Data

Creates the entity based on the data in the regression points table and leaves the Regression Data vista open.

**✔** Dismiss this Vista

Creates the entity based on the data in the regression points table and closes the Regression Data vista. Other vistas in Panorama remain open.

**?** Display Help Topic

Opens the Regression Data Vista (for Best Fit Entity Analysis) help topic.

**✕** Close

Closes the Regression Data vista and all open vistas in Panorama. Hides temporary graphics in the drawing window. Regression data and temporary graphics are saved and will reappear when the Best Fit command is reentered.

## Regression Points Table

Pt No.

Sequential number generated in the order in which the regression points were generated. Number is adjusted if regression points are added or deleted.

Exclude

Specifies the regression points to exclude from the best fit analysis.

Pass Through

Specifies a regression point which the best fit entity must pass through. Only one pass-through point may be selected for a line. Two pass-through points may be selected for an arc or parabola.

Point Northing

Regression point's Northing value. This value may be edited.

Point Easting

Regression point's Easting value. This value may be edited.

Point Station

For best fit profile entities, the regression point's station value. This value may be edited.

Point Elevation

For best fit profile entities, the regression point's elevation value. This value may be edited.

Weight

Specifies the regression point's preference in relation to other points. A higher value gives a point preference over points with lower values. This value may be edited, but it must be greater than zero.

Offset to Entity

Specifies the distance of the regression point from the best fit entity.

Northing on Entity

Specifies the Northing ordinate from the best fit entity.

Easting on Entity

Specifies the Easting ordinate from the best fit entity.

**Regression Graph**

This graph displays the regression points' relationship to the best fit entity. The red line represents the regression points and the green line represents the best fit entity. Selecting a row in the Regression Points table displays an **X** and dashed line in the graph to indicate the location of the selected point. The selected point is also highlighted in the drawing window.

**Related procedures:**

■ Creating Entities by Best Fit (page 582)

# Specify Station Range Dialog Box

Use this dialog box to specify a range of stations within a profile from which to sample data for a best fit entity regression.

Start Station

Specifies the beginning station of the range from which to take the sample. To edit the start station, enter a value or click ⬚ to select a location on the profile.

End Station

Specifies the ending station of the range from which to take the sample. To edit the end station, enter a value or click ⬚ to select a location on the profile.

**Related procedures:**

■ Creating Entities by Best Fit (page 582)

# Curve Calculator Dialog Box

Use this dialog box to calculate curve parameters from input.

**NOTE** To change the units used in the Curve Calculator dialog box, use the Ambient Settings tab of the Drawing Settings dialog box. Under Angle, change the Format value to the unit you want to use. For more information, see To specify ambient settings for a drawing (page 68).

**The following illustration shows the curve parameters used by AutoCAD Civil.**



Degree Of Curve Definition

Specifies whether the degree of curve will be based on the arc or the chord.

- **Chord Definition**: Specifies that the degree of curve is determined by the chord. Use this option if the curve is a railway curve. For a railway curve, the degree of curve is the angle at the center of a circular curve subtended by a chord of 100 units.

- **Arc Definition**: Specifies that the degree of curve is determined by the arc. Use this option if the curve is a roadway curve. For a roadway curve, the degree of curve is the central angle subtended by a circular arc of 100 units.

**The following illustration shows the degree of curve
definition for arcs and chords.**

**Degree of Curve Arc Definition**



**Degree of Curve Chord Definition**



Fixed Property

Specifies which property to hold as fixed while performing calculations.

- **Radius**: Specifies that the radius will be fixed.

- **Delta Angle**: Specifies that the delta angle will be fixed.

Degree Of Curve

Specifies the degree of curve.

Delta Angle

Specifies the delta angle of the curve. This option is not adjustible if Delta Angle is specified as the Fixed Property.

Radius

Specifies the radius of the curve. This option is not adjustible if Radius is specified as the Fixed Property.

Tangent Distance

Specifies the tangent length of the curve.

Arc Distance

Specifies the arc length of the curve.

Chord Distance

Specifies the chord length of the curve.

External Distance
    Specifies the external secant length of the curve.

Mid-ordinate Distance
    Specifies the mid-ordinate length of the curve.



    Sends the current value to the command line.



    Optionally selects an arc in the drawing and displays its values in the calculator. No edits you subsequently make in the calculator are updated in the drawing.

**Related procedures:**

■ Calculating Curve Parameters (page 593)

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# Migration Dialog Box

# 59

Use the following link to access information about the AutoCAD Civil Migration dialog box.

## Import Data from Autodesk Land Desktop Project Dialog Box

Use this dialog box to convert and import data from an Autodesk Land Desktop project into an AutoCAD Civil drawing.

**Land Desktop Project Path**

Specifies the path of the Autodesk Land Desktop project containing the data that you want to import.

Enter a valid path or click ![folder icon] to browse to the location. For example, *C:\Land Projects 2009*.

**Project Name**

Specifies the name of the project that contains the data you want to import. After the path is specified, select a project in the list.

**Alignments Site**

Specifies a site for any alignments that are being imported. Select a site from the drop-down list or click ![icon] to select an object (parcel, alignment, or grading object) contained in the site to which you want to add the new alignments.

> **NOTE** The default selection is <None>, which places the alignments in the Alignments (Centerline or Miscellaneous Alignments) collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

**Parcels Site**

Specifies a site for any parcels that are being imported. Select a site from the drop-down list or click ![icon] to select an object (parcel, alignment, or grading object) contained in the site to which you want to add the new parcels.

**Overwrite Existing Data**

Specifies whether existing data that already exists in the drawing is overwritten.

**<details area>**

Imports the specified data. Navigate to the desired data and select or clear the check boxes for one or more import options:

- **Description Keys**: Imports description keys.

- **Surfaces**: Imports surface data.

- **Alignments**: Imports alignment data.

- **Profiles**: Imports both EG and FG profile data.

- **Parcels**: Imports parcel data.

- Pipe Runs: Imports pipe data.

To iteratively import data, you can use the Apply button and check the messaging information.

For better results, import one data type at a time. To import points from Autodesk Land Desktop, use the Import Point option of the Create Points command and select External Project Point Database as the format type.

For more information about importing Autodesk Land Desktop data, see Tools for Moving Land Desktop Data into AutoCAD Civil.

# Parcels Dialog Boxes

# 60

Use the following links to access information about the Parcels dialog boxes.

## Edit Feature Settings - Parcel Dialog Box

Use this dialog box to view and change parcel-related settings.

This topic documents settings in all parcel-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the ▢ drawing icon.

■ Parcel feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ◳ parcel icon.

■ Parcel command settings are identified by the ▣ command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### ◳ Default Styles

Use these settings to specify default styles for creating and labeling parcels.

Parcel Style

Specifies the default parcel style. Click in the Value column, and click ⋯ to select a style in the Parcel Style dialog box.

Parcel Area Label Style

Specifies the default parcel area label style. Click in the Value column, and click ⋯ to select a style in the Parcel Area Label Style dialog box.

Parcel Line Label Style

Specifies the default parcel line label style. Click in the Value column, and click [...] to select a style in the Parcel Line Label Style dialog box.

Parcel Curve Label Style

Specifies the default parcel curve label style. Click in the Value column, and click [...] to select a style in the Parcel Curve Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Table Creation

**NOTE** This property group is displayed when accessing the settings from the  AddParcelCurveTable, AddParcelLineTable, AddParcelSegmentTable, or AddParcelTable commands.

Use these settings to specify defaults for creating tables with parcels.

Table Style

Specifies the style for a table. Click in the Value column, and click [...] to select a style in the Table Style dialog box.

Split Table

Specifies whether a table is split into two or more sections after a specified maximum number of rows has been met.

Maximum Number of Rows

Specifies the maximum number of rows to include per section. If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of tables to include in each stack.

Table Spacing

Specifies the spacing between tables.

Tile Direction

Specifies the direction in which the table tiles (Across or Down).

## Default Options

**NOTE** This property group is displayed when accessing the setting from the AddParcelSegmentLabels command.

Use this setting to specify defaults for creating labels with parcel segments.

Labeling Direction

Specifies the segment labeling direction (Clockwise or Counter-clockwise).

## Parcel Sizing

**NOTE** This property group is displayed when accessing the settings from the CreateParcelByLayout command.

Use these settings to specify defaults for sizing parcels.

Parcel Selection Method

Specifies whether the Pick Point Within Parcel or Select Parcel Area Label selection method is used.

Minimum Area

Specifies the minimum area for a new parcel.

Minimum Frontage

Specifies the minimum frontage (page 2474) for a new parcel.

Use Minimum Frontage At Offset

Specifies whether Use Minimum Frontage At Offset is used.

Frontage offset

Specifies the frontage offset (page 2474) for a new parcel. Must be greater than zero.

Minimum Width

Specifies the Minimum width for a new parcel. Must be greater than or equal to zero.

Minimum Depth

Specifies the minimum depth for a new parcel.

Use Maximum Depth

Specifies whether maximum depth is used.

Maximum Depth

Specifies the maximum depth for a new parcel. Must be greater than or equal to zero.

## Preview Graphics

**NOTE** This property group is displayed when accessing the settings from the CreateParcelByLayout command.

Use these settings to specify defaults for temporary parcel layout preview graphics.

Frontage Color

Specifies the default color for the frontage preview.

Frontage Offset Color

Specifies the default color for the frontage offset preview.

New Parcel Color

Specifies the default color for the new parcel preview.

Minimum Frontage Color

Specifies the default color for the minimum frontage preview.

Minimum Width Color

Specifies the default color for the minimum width preview.

Minimum Depth Color

Specifies the default color for the minimum depth preview.

## Automatic Layout

**NOTE** This property group is displayed when accessing the settings from the CreateParcelByLayout command.

Use these settings to specify defaults for automatic parcel layout.

Automatic Mode

Specifies whether the automatic mode is set for parcel layout.

Remainder Distribution

Specifies remainder from a new parcel is distributed:
- Create Parcel From Remainder

- Place Remainder In Last Parcel

- Redistribute Remainder

## ✅ Convert from Entities

**NOTE** This property group is displayed (in different forms) when accessing the settings from the CreateParcelByLayout and CreateParcelFromEntities commands.

Use these settings to specify defaults for creation of parcels.

Erase All Existing Entities

Specifies whether existing entities are erased when a new parcel is created. (CreateParcelFromEntities command only)

Auto Add Segment Label

Specifies whether segment labels are automatically added to a new parcel.

## ✅ Create Parcel Right of Way

**NOTE** This property group is displayed when accessing the settings from the CreateParcelROW command.

Use this setting to specify default offset for creation of parcel rights of way.

Offset from Alignment

Specifies the offset from the alignment for the new right of way. Enter a width in the Value column or

click  and select a width in the drawing area.

## ✅ Cleanup at Parcel Boundaries

**NOTE** This property group is displayed when accessing the settings from the CreateParcelROW command.

Use these settings to specify default offset for cleanup at parcel boundaries.

Fillet Radius at Parcel Boundary Intersections

Specifies the width of the radius of the fillet at parcel boundary intersections. Enter a width in the Value

column or click  and select a width in the drawing area.

Cleanup Method

Specifies whether the cleanup method is Fillet, Chamfer, or None.

## ✅ Cleanup at Alignment Intersections

**NOTE** This property group is displayed when accessing the settings from the CreateParcelROW command.

Use these settings to specify default offset for cleanup at alignment intersections.

Fillet Radius at Alignment Intersections

Specifies the width of the radius of the fillet at alignment intersections. Enter a width in the Value column

or click  and select a width in the drawing area.

Cleanup Method

Specifies whether the cleanup method is Fillet, Chamfer, or None.

## ✅ Parcels

---
**NOTE** This property group is displayed when accessing the settings from the CreateSite command

---

Use these settings to specify default increments for parcel numbering.

Parcel: Next Automatic Area Counter
Specifies the increment by which the next new parcel area is automatically numbered.

Parcel: Next Manual Area Counter
Specifies the increment by which the next new parcel area is manually numbered.

Parcel Line: Next Automatic Tag Counter
Specifies the increment by which the next new parcel line tag is automatically numbered.

Parcel Line: Next Manual Tag Counter
Specifies the increment by which the next new parcel line tag is manually numbered.

Parcel Curve: Next Automatic Tag Counter
Specifies the increment by which the next new parcel curve tag is automatically numbered.

Parcel Curve: Next Manual Tag Counter
Specifies the increment by which the next new parcel curve tag is manually numbered.

## ✅ Alignment

---
**NOTE** This property group is displayed when accessing the settings from the CreateSite command

---

Use these settings to specify default increments for alignment tag numbering.

Alignment Spiral: Next Manual Tag Counter
Specifies the increment by which the next new alignment spiral tag is manually numbered.

Alignment Spiral: Next Automatic Tag Counter
Specifies the increment by which the next new alignment spiral tag is automatically numbered.

Alignment Curve: Next Manual Tag Counter
Specifies the increment by which the next new alignment curve tag is manually numbered.

Alignment Curve: Next Automatic Tag Counter
Specifies the increment by which the next new alignment curve tag is automatically numbered.

Alignment Line: Next Manual Tag Counter
Specifies the increment by which the next new alignment line tag is manually numbered.

Alignment Line: Next Automatic Tag Counter
Specifies the increment by which the next new alignment line tag is automatically numbered.

## ✅ Feature Line

---
**NOTE** This property group is displayed when accessing the settings from the CreateSite command

---

Use this setting to specify defaults for feature line style priority.

Feature Line Style Priority

Specifies the default feature line style priority. Click in the Value column, and click ··· to select a style in the Options Tab (Feature Line Site Properties Dialog Box) (page 1915).

### ⊞ Parcel Analysis

**NOTE** This property group is displayed when accessing the settings from the ExportParcelAnalysis command.

Use these settings to specify defaults for parcel analysis.

Analysis Type
   Specifies whether Inverse Analysis or Mapcheck Analysis is used.

Enable Mapcheck Across Chord
   Specifies whether mapcheck across chord is enabled or disabled.

Process Counter-clockwise
   Specifies whether segments will be processed in counter-clockwise or clockwise order.

**Related procedures:**

■  Editing Parcel Settings (page 824)

■  Editing Parcel Command Settings (page 825)


# Parcel Styles Dialog Box

Use this dialog box to edit settings for parcel styles.


## Information Tab (Parcel Styles Dialog Box)

Use this tab to change a parcel style name or description, or to review details, such as when the parcel style was most recently modified.

Name
   Specifies the name of the parcel style.

Description
   Specifies the description of the parcel style. Enter an optional description.

Created By
   Displays the Windows login name of the person who created the parcel style.

Date Created
   Displays the date and time the parcel style was created.

Last Modified By
   Displays the Windows login name of the person who last modified the parcel style.

Date Modified
   Displays the date and time the parcel style was modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■  Parcel Styles (page 828)


## Design Tab (Parcel Styles Dialog Box)

Use this tab to change parcel style settings for fill distance or parcel name template (naming convention).

The fill distance defines the width of the filled area inside the parcel boundary. If the fill distance is observed, the pattern fill is applied to the boundary area only.

Observe Fill Distance
Specifies whether the fill distance is observed. Select the check box to apply the fill to the boundary area only; otherwise, the fill is applied to the entire parcel.

Fill Distance
Specifies the width of the boundary area.

Parcel Name Template
Displays the parcel name template. Click  to open the Name Template dialog box (page 1806). Select or enter values for the parcel name format.

**Related procedures:**

■ Parcel Styles (page 828)

## Display Tab (Parcel Styles Dialog Box)

Use this tab to specify parcel style settings which affect the following parcel segments or area fill: Layer, Color, Linetype, LT Scale, Lineweight, and Visibility. For area fill, you can specify additional settings for a hatch: Pattern, Angle, or Scale.

For more information, see Display Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■ Parcel Styles (page 828)

## Summary Tab (Parcel Styles Dialog Box)

Use this tab to review Information or Design settings for a parcel style. You can copy this information to a spreadsheet.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

**Related procedures:**

■ Parcel Styles (page 828)

# Create Parcels - Layout Dialog Box

Use this dialog box to review settings that are applied to new parcels.

Site
Specifies the site for the new parcels.

Parcel Style
Specifies the parcel style for the new parcels.

**Layers**

Parcel Layer
Specifies the layer for the new parcels.

Parcel Segment Layer
   Specifies the layer for the new parcel segments.

**Label Styles**

Area Label Style
   Specifies the area label style for new parcels.

Line Segment Label Style
   Specifies the line segment label style for new parcels.

Curve Segment Label Style
   Specifies the curve segment label style for new parcels.

Automatically Add Segment Labels
   Specifies whether to add segment labels while segments are created.

Erase Existing Entities
   When creating parcels from objects, specifies whether to erase original objects after parcels are created.
   When the Xref option is selected, this option is disabled.

**Related procedures:**

■

■

# Parcel Layout Parameters Dialog Box

Use this dialog box to change the properties of parcel segments.

Attributes displaying as black text can be edited. Areas that are dimmed (gray) are not available.

---

**NOTE** The constraint definition for each parcel subentity determines which parameters are displayed in the Parcel Layout Parameters window.

---

You cannot dock the Parcel Layout Parameters dialog box. You can pin it, and resize the window.

Use the horizontal and vertical scroll bars when the list is longer or wider than the window.

**General**

Type
   Specifies either Line or Curve as the entity type.

Constraint 1
   Specifies either Fixed, Free, or Floating as the general constraint type.

Constraint 2
   Specifies a detailed description of the entity constraints.

**Geometry (Line)**

Length
   Specifies the line length.

Direction
   Specifies the direction of the line.

**Start Station**

Specifies the starting station of the line for the parcel segment.

**End Station**

Specifies the end station of the line for the parcel segment.

**Start Point**

Specifies the XY coordinates for the start point of the line.

**End Point**

Specifies the XY coordinates for the endpoint of the line.

**Pass Through Point 1**

Specifies the XY coordinates of the first user-specified pass-through point.

**Pass Through Point 2**

Specifies the XY coordinates of the second user-specified pass-through point.

## Geometry (Curve)

**Length**

Specifies the length of the curve entity.

**Radius**

Specifies the radius of the curve entity.

**Delta Angle**

Specifies the included angle of the solved portion of the curve.

**Start Station**

Specifies the starting station of the curve for the solved portion of the alignment. Unsolved curves do not display a start station.

**End Station**

Specifies the end station of the curve for solved portions of the alignment. Unsolved curves do not display an end station.

**Start Direction**

Specifies the direction of the tangent coming into the curve.

**End Direction**

Specifies the direction of the tangent going out of the curve.

**Start Point**

Specifies the XY coordinates of the start point of the curve.

**End Point**

Specifies the XY coordinates of the endpoint of the curve.

**Center Point**

Specifies the XY coordinates of the center point.

**Pass Through Point1**

Specifies the XY coordinates of the first user-specified pass-through point.

**Pass Through Point2**

Specifies the XY coordinates of the second user-specified pass-through point.

**Pass Through Point3**

Specifies the XY coordinates of the third user-specified pass-through point.

Chord Length

    Specifies the distance along the line joining the beginning of the curve (PC) and the end of the curve (PT).

Chord Direction

    Specifies the direction along the line joining the beginning of the curve (PC) and the end of the curve (PT).

Mid-Ordinate

    Specifies the distance from the midpoint of the curve, perpendicular to the midpoint of the chord.

External Tangent

    Specifies the distance from the beginning of the curve (PC) to the PI or from the PI to the end of the curve (PT).

External Secant

    Specifies the radial difference from the PI to the midpoint of the curve.

PI Included Angle

    Specifies the included angle between the incoming and outgoing tangents of the curve.

**Related procedures:**

■  Editing Parcels by Layout (page 820)


# Edit Parcel Properties Dialog Box

Use this dialog box to edit the properties of a group of parcels.

After specifying the parcel style, area label styles, and user-defined properties for one parcel, you can apply them to all selected parcels.

**General**

Left and Right Arrows

    Displays the name of the first selected parcel. Use the arrow keys to view or edit the properties of other parcels.

Parcel Style

    Specifies the parcel style. Editing a style only affects the currently selected parcel. Click  to apply the style to all selected parcels.

Area Selection Label Style

    Specifies the Area Selection Label Style that is applied to the parcel. Click  to apply the area selection label style to all selected parcels.
    Specifies the current area label styles. Click to apply these area label styles to all selected parcels. If you try to add a style that is already in the list, you are prompted to select another style. Click  to add an area style label.

    Click  to delete one or more area label styles. If you do not select at least one style in the list box, you are prompted to select one. If there is only one area label style and you try to delete it, an error message displays.

Area Label Styles

    Specifies the Area Label Styles that are applied to the parcel. Click  to apply these area label styles to all selected parcels.

Specifies the current area label styles. Click to apply these area label styles to all selected parcels. If you try to add a style that is already in the list, you are prompted to select another style. Click ✚ to add an area style label.

Click ✖ to delete one or more area label styles. If you do not select at least one style in the list box, you are prompted to select one. If there is only one area label style and you try to delete it, an error message displays.

User Defined Properties

Lists user-defined properties in the parcel that you are editing. Only the value field can be edited. Click ⟳ to apply the selected property to all selected parcels. If you do not select a property in the list you are prompted to do so. Each parcel must have its own unique number, a parcel number cannot be applied to other parcels.

**Related procedures:**

■ Editing Parcels by Layout (page 820)

■ Editing Parcel Properties (page 825)


# Parcel Layout Tools

Use this toolbar to access tools for working with parcel segments and setting parameters for parcel size.

Create Parcel

Opens the Create Parcels - Layout dialog box (page 2005) where you can change default parcel settings during a layout session.

## Lot Line Tools

Add Fixed Line - Two Points

Draws a lot line as a line segment. Click a starting point and an endpoint in the drawing.

Add Fixed Curve - Three Points

Draws a lot line as a curve segment. Define a starting point, pass-through point, and endpoint.

Add Fixed Curve - Two Points And Radius

Draws a lot line as a curve segment. Define a starting point, radius value, curve direction, and endpoint.

Draw Tangent-Tangent With No Curves

Draws a connected series of lot line segments. Click a sequence of points.

## Parcel Sizing Tools

These tools precisely control the position of individual lot lines while respecting user-defined parameters. The Parcel Sizing Tools create or edit attached lot lines.

Use either the Slide Line Create or Swing Line Create command in automatic mode to create a series of evenly sized adjacent parcels. For manual layout, specify the new location for a lot line by picking a point or dragging the line using OSNAP.

When Parcel Sizing parameters or Automatic Layout options are selected or specified, a corresponding preview graphic displays at the bottom of the Parcel Layout Tools dialog box.

**Slide Line - Create**

Creates one or more new lot lines that you define with start and end points along the frontage, and optionally, an angle relative to the frontage or an absolute direction for the lot line. Relative angles are measured in + degrees, from 0 (toward the endpoint) through 180 (toward the start point). Note that 90 degrees is perpendicular/radial to the frontage definition. Define the direction by means of azimuth, bearing, or two points in the drawing.

**Slide Line - Edit**

Moves a lot line. You can retain or change the line's frontage angle or absolute direction.

**Swing Line - Create**

Creates a lot line defined with start and end points along the frontage and a fixed swing point on the opposite side of the parcel. Adjust the size of the parcel by swinging the lot line to intersect a different point along the frontage, subject to a minimum area and frontage limit.

**Swing Line - Edit**

Moves a lot line by swinging it from one end. Select which end to use as the swing point.

**Free Form Create**

Creates a new lot line. Define an attachment point and a bearing, azimuth, or second attachment point.

### PI Editing Tools

**Insert PI**

Inserts a vertex at the point you click on a parcel segment.

**Delete PI**

Deletes a vertex that you select on a parcel segment, and redraws the lot line between the vertices on either side.

**Break Apart a PI**

Separates end points at the vertex you select. Specify a separation distance.

**NOTE** Breaking a PI does not delete or merge parcels, as deleting segments does. It merely makes parcels incomplete. Affected segments revert to geometry elements, and parcel labels disappear. Geometry elements become parcels again if you reconnect loose vertices to make closed figures.

### Other Tools

**Delete Sub-Entity**

Deletes a parcel subentity, such as a lot line or curve. If you delete a subentity that is not shared by another parcel, the entire parcel is deleted. If you delete a shared subentity, the two parcels that shared it are merged.

**Parcel Union**

Joins two adjacent parcels. The first parcel you select determines the identity and properties of the joined parcel.

**Pick Sub-Entity**

Selects a parcel subentity for display in the Parcel Layout Parameters dialog box. Click Sub-entity Editor before you click this button.

**Sub-Entity Editor**

Opens the Parcel Layout Parameters dialog box (page 2006) where you can review or edit attributes of the selected parcel subentity.

**Undo**

Cancels the previous command.

**Redo**

Repeats the cancelled command.

**Expand the Toolbar**

Opens a property sheet where you can change parameters for new parcel sizing, automatic layout, and manual layout.

## Parcel Sizing Parameters

When a parcel sizing parameter is selected, a graphic displays to illustrate how the parameter affects the result of the current parcel sizing command.

Minimum Area

Specifies the minimum area criteria for new parcels. This value can be changed at anytime during the current command and must be greater than zero.

**NOTE** If the minimum frontage criteria is met, but the minimum area has not, the parcel frontage will be increased until the minimum area is obtained.



Minimum Frontage

Specifies the minimum frontage criteria for new parcels.

**NOTE** If the minimum area criteria is met, but the minimum frontage has not, the parcel area will be increased until the minimum frontage is obtained.

**Use Minimum Frontage At Offset**

Enables/disables the use of the minimum frontage at offset during the parcel sizing commands. This value can be changed at anytime during the current command.



**Frontage Offset**

Specifies the default value for the frontage offset.



**Minimum Width**

Sets the default value for Minimum width. This value must be greater than or equal to zero.



**Minimum Depth**

Specifies the minimum depth criteria used to size new or edit existing parcels. The minimum depth starts at the mid-point of the resulting frontage of the new or edited parcel and is oriented normal (perpendicular) to the intersected frontage segment.

**NOTE** If the minimum area and frontage criteria are met, but not the minimum depth, the frontage and area are increased until the minimum depth is obtained. If no solution is found, a command prompt displays.

**Use Maximum Depth**

Specifies whether the maximum depth criteria is used when creating or editing parcels.



**Maximum Depth**

Specifies the maximum depth criteria for a new or edited parcel.

**NOTE** If the minimum area and frontage criteria are met, and the maximum depth is exceeded, a command prompt displays that no solution is found.

**Multiple Solution Preference**

In certain situations, multiple valid solutions may occur during the creation of new parcels. When multiple valid solutions occur, you can specify which solution is displayed.

Use Shortest Frontage: Specifies that the solution with the shortest frontage is displayed.



Use Smallest Area: Specifies that the solution with the smallest area is displayed.

## Automatic Layout

Automatic Mode

Specifies whether automatic mode is enabled for parcel sizing commands.
On: Enables automatic mode.

Off: Disables automatic mode.

Remainder Distribution

Specifies the method of redistributing the remainder.
Create Parcel From Remainder: Creates a new parcel from the remaining area.

Place Remainder in Last Parcel: Places the remaining area in the last parcel.

Redistribute Remainder: Redistributes the remaining area evenly to every parcel.



**Related procedures:**

■ Creating Parcels by Layout (page 815)

■ Editing Parcels by Layout (page 820)

■ Editing Subdivided Parcels (page 820)

# Parcel Properties Dialog Box

Use this dialog box to change the properties of a single parcel.

You can edit parcel name, description, object style, label style, analysis, and user-defined properties.

# Information Tab (Parcel Properties Dialog Box)

Use this tab to change a parcel, name, description, or style.

Name
Specifies the name of the parcel.

Description
Specifies the description of the parcel. Enter an optional description.

Object Style
Specifies the parcel style applied to the parcel.

Use Name Template In Parcel Style
Specifies whether the parcel name is based on the parcel style name template (page 1806).
Selected: The parcel name is (re)set based on the parcel style name template. The Name field is disabled for manual editing (the Name column is also disabled in List View (page 83)).
Cleared: Manually enter a parcel name in the Name field (You can also edit the Name column in the List View (page 83)).

**NOTE** This manually entered name is not automatically updated with subsequent changes to the parcel number.

Show Tooltips
Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

**Related procedures:**

■ Parcel Properties (page 825)

## Composition Tab (Parcel Properties Dialog Box)

Use this tab to change a parcel area label style, or to view its area or perimeter.

Area Selection Label Style

Specifies the area selection label style that is applied to the parcel.

### Parcel Statistics

Area

Displays the area of the parcel.

Perimeter

Displays the perimeter of the parcel.

### Related procedures:

■ Editing Parcel Properties (page 825)

## Analysis Tab (Parcel Properties Dialog Box)

Use this tab to change parcel analysis type, point of beginning, or segment process-order, or to view analysis results.

### Analysis

NOTE Analysis results are displayed in the Result Area, which is the lower portion of the dialog box.

Inverse Analysis

Specifies that Inverse analysis type is used for the parcel.
Inverse analysis provides a sequential report of directions and distances, start and end coordinates, curve data for each parcel segment starting at a specified Point of Beginning (POB), and parcel area.

Mapcheck Analysis

Specifies that Mapcheck analysis type is used for the parcel.
Mapcheck analysis provides the same information as Inverse analysis, except that all start and end coordinates for each parcel segment are computed relative to the coordinates of the POB and the previous segment, using the labeled precision of the parcel segment's direction and distance/curve data. Therefore, error is introduced and accumulated so that the coordinates of the last segment will not equal that of the POB. This is termed 'error of closure'. Mapcheck reports check the plotted drawing for omissions of segment labels to avoid errors that may be introduced into legal documents, such as deed descriptions.

Enable Mapcheck Across Chord

This setting is available only if Mapcheck Analysis is selected.
If mapcheck across chord is enabled, the Mapcheck traverse is calculated for curve segments using their chord length. Otherwise, it is calculated using their curve length.

### Calculation Settings

Point Of Beginning

Specifies the parcel boundary's initial node (start point of its initial segment).

 Point of Beginning Selector

Selects a new point of beginning. Click the button and click a parcel node in the drawing.

Process Segment Order Counter Clockwise

Specifies that segments are processed in counter-clockwise order.

**Result Area**

(Not labeled) Displays analysis results in the lower portion of the dialog box.

**Related procedures:**

■ Parcel Properties (page 825)

# User Defined Properties (Parcel Properties Dialog Box)

Use this tab to review or change user defined parcel properties.

Parcel Number
   Specifies the number assigned to the parcel.

Parcel Address
   Specifies the address of the parcel.

Parcel Tax ID
   Specifies the Tax ID number assigned to the parcel.

**NOTE** The user-defined property classification cannot be defined here. It is set in the Site Parcel Properties dialog.

**Related procedures:**

■ Parcel Properties (page 825)

■ Editing Parcel Properties (page 825)

# Site Parcel Properties Dialog Box

Use this dialog box to change properties of a site parcel.

# Composition Tab (Site Parcel Properties Dialog Box)

Use this tab to change a site parcel's label style, to view its area or perimeter, or to change the display order of parcels that it contains, including the site parcel itself.

Site Parcel Style
   Specifies the style that is applied to the site parcel.

Site Area Label Style
   Specifies the area label style that is applied to the site parcel.
   Specifying <none> hides the site parcel's area label.

Parcel Style Display Order
   Specifies the display order of parcels within the site, including the site parcel itself, according to the styles that are assigned to them.
   The style highest in this list if first in the display order. Where parcels of two different styles share a common segment, the segment appears in the style that is higher in the display order.
   To change a style's list position, select it. Click an Up or Down Arrow. You can also drag a style up or down to a new position.

Parcel Area
   Displays the area of the site parcel.

Parcel Perimeter

Displays the perimeter of the site parcel.

User-defined property classification

Specifies the user-defined property classification used to group user-defined properties.

**Related procedures:**

- Using User-Defined Property Classifications with Parcels (page 853)

- Changing Parcel Display Order (page 827)

- Hiding or Showing Parcel Labels (page 842)

## Analysis Tab (Site Parcel Properties Dialog Box)

Use this tab to change a site parcel's analysis type, its point of beginning, its segment process-order, or to view analysis results.

### Analysis

**NOTE** Analysis results are displayed in the Result Area in the lower portion of the dialog box.

Inverse Analysis

Specifies that Inverse analysis type is used for the parcel.
Inverse analysis provides a sequential report of directions and distances, start and end coordinates, curve data for each parcel segment starting at a specified Point of Beginning (POB), and parcel area.

Mapcheck Analysis

Specifies that Mapcheck analysis type is used for the parcel.
Mapcheck analysis provides the same information as Inverse analysis, except that all start and end coordinates for each parcel segment are computed relative to the coordinates of the POB and the previous segment, using the labeled precision of the parcel segment's direction and distance/curve data. Therefore, error is introduced and accumulated so that the coordinates of the last segment will not equal that of the POB. This is termed 'error of closure'. Mapcheck reports check the plotted drawing for omissions of segment labels to avoid errors that may be introduced into legal documents, such as deed descriptions.

Enable Mapcheck Across Chord

Enables map check across chord. This setting is available only if Mapcheck Analysis is selected.
If mapcheck across chord is enabled, the Mapcheck traverse is calculated for curve segments using their chord length. Otherwise, it is calculated using their curve length.

### Calculation Settings

Point of Beginning

Specifies the site parcel boundary's initial node (start point of its initial segment).

 Point of Beginning Selector

Selects a new point of beginning. Click the button and click a site parcel node in the drawing.

Process Segment Order Counter Clockwise

If selected, segments are processed in counter-clockwise order.

### Result Area

(Not labeled) Displays analysis results in the lower portion of the dialog box.

# Export Parcel Analysis Dialog Box

Use this dialog box to export multiple inverse or mapcheck parcel analysis reports to a chosen location.

**NOTE** To perform a Mapcheck Analysis on parcel labels, click General menu ➤ Mapcheck Analysis. For more information, see Performing a Mapcheck Analysis (page 1608).

Destination File

Specifies the file name and the location to export the analysis report to. Enter the name and file path into the edit box or click to browse to a location. The edit box will default to the last used location.

## Analysis Type

Inverse Analysis

Specifies that Inverse analysis type is used for the parcel.
Inverse analysis provides a sequential report of directions and distances, start and end coordinates, curve data for each parcel segment starting at a specified Point of Beginning (POB), and parcel area.

Mapcheck Analysis

Specifies that Mapcheck analysis type is used for the parcel.
Mapcheck analysis provides the same information as Inverse analysis, except that all start and end coordinates for each parcel segment are computed relative to the coordinates of the POB and the previous segment, using the labeled precision of the parcel segment's direction and distance/curve data. Therefore, error is introduced and accumulated so that the coordinates of the last segment will not equal that of the POB. This is termed 'error of closure'. Mapcheck reports check the plotted drawing for omissions of segment labels to avoid errors that may be introduced into legal documents, such as deed descriptions.

Enable Mapcheck Across Chord

Enables map check across chord. This setting is available only if Mapcheck Analysis is selected.
If mapcheck across chord is enabled, the Mapcheck traverse is calculated for curve segments using their chord length. Otherwise, it is calculated using their curve length.

Process Segment Order Counter Clockwise

If selected, segments are processed in counter-clockwise order.

**Related procedures:**

■   Exporting a Parcel Inverse or Mapcheck Report (page 830)

# Renumber/Rename Parcels Dialog Box

Use this dialog box to renumber and/or rename parcels.

Parcel names using the parcel style name template (page 1806) (with the parcel number property) are updated automatically.

The default Starting Number is inherited from Parcel: Next Manual Area Counter on the Numbering tab of the Site Properties dialog box for the site you are working with. After you renumber a set of parcels, the site retains the next available number.

Site Name

Specifies the site in which you are renumbering parcels.

Renumber

Specifies whether to renumber the selected parcels. The names of the parcels may or may not be dynamically changed, depending on the other settings.

Starting Number

Specifies the next number that will be used for the selected parcel. If a duplicate number occurs, the current parcel's number and the duplicate parcel's number are swapped.

Increment Value

Specifies the difference between numbers in the sequence.

Use Name Template In Parcel Style

Specifies whether the names of the selected parcels are obtained from the parcel style name template. Selected: The parcel name is (re)set based on the parcel style name template.
Cleared: The parcel number will change and, depending on the current properties of the selected parcel, the parcel name may update.

Rename

Specifies whether the selected parcels will be renamed.

Specify The Parcel Name

Manually enter the parcel name. Note that this hard-coded name is no longer the "dynamic" name that updates if parcels are renumbered.

Click  to open the Name Template dialog box (page 1806). Select or enter values for the parcel name format. The parameters specified are returned and displayed in the Specify The Parcel Name field.

Use Name Template In Parcel Style

Specifies that the names of the selected parcels will be reset using the specified parcel style name template.

**Related procedures:**

■  Renumbering Parcels (page 850)

■  Numbering Parcel Segments (page 852)

# Create Right of Way Dialog Box

Use this dialog box to offset parcel boundaries from both sides of an alignment.

## Create Parcel Right Of Way

Offset From Alignment

Specifies the offset distance of each side of the right-of-way from the alignment.

## Cleanup At Parcel Boundaries

Fillet Radius At Parcel Boundary Intersections

Specifies the fillet radius or chamfer distance at boundary intersections.

Cleanup Method

None: Specifies that no cleanup is used at parcel boundary intersections.
Fillet: Specifies that a fillet is used at parcel boundary intersections. A fillet connects the parcel boundaries with an arc that is tangent to the boundaries and has a radius specified by Fillet Radius At Parcel Boundary Intersections.
Chamfer: Specifies that a chamfer is used at parcel boundary intersections. A chamfer connects the parcel boundaries with an angled line at a chamfer distance specified by Fillet Radius At Parcel Boundary Intersections.

## Cleanup At Alignment Intersections

Fillet Radius At Alignment Intersections

Specifies the fillet radius or chamfer distance at alignment intersections.

Cleanup Method

None: Specifies that no cleanup is used at alignment boundary intersections.

Fillet: Specifies that a fillet is used at alignment intersections. A fillet connects the alignment boundaries with an arc that is tangent to the alignments and has a radius specified by Fillet Radius At Alignment Intersections.

Chamfer: Specifies that a chamfer is used at alignment intersections. A chamfer connects the alignments with an angled line at a chamfer distance specified by Fillet Radius At Alignment Intersections.

**Related procedures:**

■ Creating a Right of Way (page 822)

# Parcel Move To/Copy To Site Dialog Boxes

Use these common dialog boxes to move or copy parcels from one site to another.

The contents of an entire site, including alignments, grading groups, or parcels, can be moved or copied together. For more information, see Move To Site dialog box (page 2327) and Copy To Site dialog box (page 2328).

**Related procedures:**

■ Move Objects to a Site (page 726)

■ Copy Objects to a Site (page 727)

# Pipe Networks Dialog Boxes

# 61

Use the following links to access information about the Pipe Networks dialog boxes.

## Create Pipe Network By Layout Dialog Box

Use this dialog box to enter the initial pipe network creation parameters such as a name, optional description, parts list, layer, and more.

Also, you can select a surface and/or an alignment to associate with the pipe network, as well as styles for structure and pipe labels.

Network Name

Specifies the name of the pipe network. Each pipe network must have a unique name.



Opens the Name Template (page 1806) dialog box, where you can modify the pipe network naming template.

Network Description

Specifies an optional description for the pipe network.

Network Parts List

Specifies the parts list associated with this pipe network.

Layers

Opens the Pipe Network Layers (page 2063) dialog box, where you can assign various views of the pipe network components to specific layers in the drawing.

Surface Name

Specifies the surface associated with this pipe network.

Alignment Name

Specifies the alignment associated with this pipe network.

### Structure Label Style

Style List

Displays the current style. Click the arrow to display the structure label styles in the drawing.

Style Selection

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box, where you can preview the style and creation information.

**Pipe Label Style**

Style List

Displays the current style. Click the arrow to display the pipe label styles in the drawing.

Style Selection

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box, where you can preview the style and creation information.

**Related procedures:**

■ Creating a Pipe Network Using the Layout Tools (page 1192)

# Create Pipe Network From Object Dialog Box

Use this dialog box to create a pipe network from a drawing entity such as a line, polyline, arc, or feature line.

Using this feature, you can select one of the following entity types in a drawing and create a pipe network from it: line, 2D or 3D polylines, 2D spline polyline, arc, feature line.

You are prompted to enter the initial pipe network creation parameters, including a pipe network name, description, parts list, layers, and more. Also, you can select a surface and/or an alignment to associate with the pipe network.

When using this feature, if the selected parts list has a mismatch with the selected catalog, the OK button is disabled, as well as the selector for the structure, where the structure types within the parts list have a mismatch with the currently selected catalog. An error message will also be displayed informing you that parts in the selected part list cannot be found in the current catalog. You are prompted to select a different part list or to select the correct catalog.

Network Name

Specifies the name of the pipe network. Each pipe network must have a unique name.



Opens the Name Template (page 1806) dialog box, where you can modify the pipe network naming template.

Network Description

Specifies an optional description for the pipe network.

Network Parts List

Specifies the parts list associated with this pipe network.

Pipe To Create

Specifies the part that will be used to represent pipes in this pipe network.

Structure To Create

Specifies the part that will be used to represent structures in this pipe network.

Layers

Opens the Pipe Network Layers (page 2063) dialog box, where you can assign various views of the pipe network components to specific layers in the drawing.

Surface Name

Specifies the surface associated with this pipe network.

Alignment Name

Specifies the alignment associated with this pipe network.

Erase Existing Entity

Specifies whether the entity (object) you selected will be erased automatically when you create the pipe network. Select the check box to erase the entity automatically when you create the pipe network.

NOTE This option is disabled when you select an object from an XREF.

Use Vertex Elevations

When this option is selected, the elevations of any vertexes along the selected entity are used to set the elevations of the pipes created in the network. For 3D entity types, this option determines if the 3D vertex elevations are honored by the pipes created. When selecting this option, rules are not applied, since they would likely be in conflict with the vertex elevations.

**Related procedures:**

■ Creating Pipe Networks from Objects (page 1194)

# Create Alignment From Network Dialog Box

Use this dialog box to create an alignment by selecting parts within a pipe network.

You can optionally create a profile and profile view as you create the new alignment.

Site

Specifies a site for the alignment. Either select a site from the Site list or click [icon] to select an object in the drawing. The alignment and the object you select in the drawing are associated with the same site.

NOTE The default Site selection is <None>, which places the alignment in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

Name

Specifies the name of the alignment. Each alignment style must have a unique name.

Description

Specifies an optional description for the alignment.

Starting Station

Specifies the start station for the alignment.

**Alignment Style**

Style List

Displays the current style. Click the arrow to display the alignment styles in the drawing.

Style Selection

Specifies the style options. You can create a new style, copy or edit the current style selection, or pick a style from drawing.

 Style Detail

> Opens the Style Detail dialog box. Preview the style and creation information.

Alignment Layer

> Displays the layer on which the alignment object will be created. To change the layer click the Object Layer icon and select a layer.

 Object Layer

> Opens the Object Layer dialog box. Click  to create a new layer on which the alignment will be drawn.

Alignment Label Set

> Lists the alignment Label Set styles in the drawing. Use the Selection icon to change the current style or create a new style. The Style Detail icon previews the current style.

Create Profile and Profile View

> Specifies whether a profile and a profile view are created for the alignment. If you select this check box, the Create Profile View dialog box (page 2145) is displayed.

**Related procedures:**

- Creating Alignments from Pipe Network Parts (page 1196)

- Creating Alignments (page 887)

# Edit Feature Settings - Pipe Network Dialog Box

Use this dialog box to view and change pipe network-related settings.

This topic documents settings in all pipe network-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the  drawing icon.

- Pipe network feature settings are listed near the top of this dialog box, after the General property group, and are identified by the  pipe network icon.

- Pipe network command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

##  Default Styles

Use these settings to establish the default styles assigned to pipe network components.

Interference Default Style

> Specifies the default style for interference in the pipe network. Click in the Value column, and click  to select a style in the Interference Default Style dialog box.

Structure Default Style

> Specifies the default style for structures in the pipe network.Click in the Value column, and click  to select a style in the Structure Default Style dialog box.

Pipe Default Style

Specifies the default style for pipes in the pipe network. Click in the Value column, and click ⊡ to select a style in the Pipe Default Style dialog box.

Interference Render Material

Specifies the default interference render material in the pipe network. Click in the Value column, and click ⊡ to select a style in the Interference Render Material dialog box.

Structure Plan Label Style

Specifies the default style for structure labels that are displayed in plan view. Click in the Value column, and click ⊡ to select a style in the Structure Plan Label Style dialog box.

Pipe Plan Label Style

Specifies the default style for pipe labels that are displayed in plan view. Click in the Value column, and click ⊡ to select a style in the Pipe Plan Label Style dialog box.

Structure Profile Label Style

Specifies the default style for structure labels that are displayed in a profile view. Click in the Value column, and click ⊡ to select a style in the Structure Profile Label Style dialog box.

Pipe Profile Label Style

Specifies the default style for pipe labels that are displayed in profile view. Click in the Value column, and click ⊡ to select a style in the Pipe Profile Label Style dialog box.

Structure Section Label Style

Specifies the default style for structure labels that are displayed in a section view. Click in the Value column, and click ⊡ to select a style in the Structure Section Label Style dialog box.

Pipe Section Label Style

Specifies the default style for pipe labels that are displayed in section view. Click in the Value column, and click ⊡ to select a style in the Pipe Section Label Style dialog box.

Render Material

Specifies the default render material in the pipe network. Click in the Value column, and click ⊡ to select a style in the Render Material dialog box.

Default Parts List

Specifies the default parts list for the pipe network. Click in the Value column, and click ⊡ to select a style in the Default Parts List dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### 🏃 Default Name Format

Use these settings to specify the default name formats for pipe network components (pipe network, pipes, structures). Click in the Value column, and click ⊡ to make changes in the Name Template dialog box (page 1806).

### 🏃 Default Rules

Use these settings to establish the default rule set assigned to pipes and structures that are added to the pipe network.

Structure Default Rules

Specifies the default rule set assigned to structures. Click in the Value column, and click ⊡ to select a set in the Structure Default Rules dialog box.

Pipe Default Rules

Specifies the default rule set assigned to pipes. Click in the Value column, and click ⊡ to select a set in the Pipe Default Rules dialog box.

## Storm Sewers Migration Defaults

Use these settings to establish the defaults for part matching, and the default parts list, used when migrating pipe network data between the Storm Sewers Extension and AutoCAD Civil.

Part Matching Defaults

Specifies the defaults used for part matching when migrating pipe network data between the Storm Sewers Extension and AutoCAD Civil. Click in the Value column, and click ⊡ to display the Part Matchup Settings dialog box (page 2079) for importing and exporting. For more information, see Setting Part Matching Defaults for Migration (page 1276).

Parts List Used For Migration

Specifies the default parts list used when migrating pipe network data between the Storm Sewers Extension and AutoCAD Civil. Click in the Value column, and click ⊡ to select a parts list. For more information, see Setting the Default Parts List For Migration (page 1277).

## Default Profile Label Placement

Use these settings to establish the default placement of profile view labels associated with the pipe network.

Dimension Anchor Option for Pipes

Specifies the location of the dimension line anchor for profile view labels for pipes:

- Fixed

- Above

- Below

- Graph View Top

- Graph View Bottom

Dimension Anchor Elevation Value for Pipes

Specifies the default elevation of the dimension line anchor for profile view labels for pipes. Enter an elevation in the Value column or click ▦ and select an elevation in the drawing area.

Dimension Anchor Plot Height Value for Pipes

Specifies the default plot height of the dimension line anchor for profile view labels for pipes. Enter a height in the Value column.

Dimension Anchor Option for Structures

Specifies the location of the dimension line anchor for profile view labels for structures:

- Fixed

- Above

- Below

- Graph View Top

- Graph View Bottom

Dimension Anchor Elevation Value for Structures

Specifies the default elevation of the dimension line anchor for profile view labels for structures. Enter an elevation in the Value column or click  and select an elevation in the drawing area.

Dimension Anchor Plot Height Value for Structures

Specifies the default plot height of the dimension line anchor for profile view labels for structures. Enter a height in the Value column.

Structure Label Placement

Specifies the default location of structure labels:.

- At Top Of Structure

- At Middle Of Structure

- At Bottom Of Structure

## Default Section Label Placement

Use these settings to establish the default placement of section view labels associated with the pipe network.

Dimension Anchor Option for Pipes

Specifies the location of the dimension line anchor for section view labels for pipes:

- Fixed

- Above

- Below

- Graph View Top

- Graph View Bottom

Dimension Anchor Elevation Value for Pipes

Specifies the default elevation of the dimension line anchor for section view labels for pipes. Enter an elevation in the Value column or click  and select an elevation in the drawing area.

Dimension Anchor Plot Height Value for Pipes

Specifies the default plot height of the dimension line anchor for section view labels for pipes. Enter a height in the Value column.

Dimension Anchor Option for Structures

Specifies the location of the dimension line anchor for section view labels for structures:

- Fixed

- Above

- Below

- Graph View Top

- Graph View Bottom

Dimension Anchor Elevation Value for Structures

Specifies the default elevation of the dimension line anchor for section view labels for structures. Enter an elevation in the Value column or click  and select an elevation in the drawing area.

Dimension Anchor Plot Height Value for Structures

Specifies the default plot height of the dimension line anchor for section view labels for structures. Enter a height in the Value column.

Structure Label Placement

Specifies the default location of structure labels:

- At Top Of Structure

- At Middle Of Structure

- At Bottom Of Structure

Pipe Section Label Placement

Specifies the default location of pipe labels displayed in section views:

- At Top Of Structure

- At Middle Of Structure

- At Bottom Of Structure

## Table Creation

**NOTE** This property group is displayed when accessing the settings from the AddNetworkPipeTable and AddNetworkStructTable commands.

Use these settings to establish the defaults when you add a network pipe or structure table.

Table Style

Specifies the style for a table. Click in the Value column, and click ⋯ to select a style in the Table Style dialog box.

Split Table

Specifies whether a table is split into two or more sections after a specified maximum number of rows has been met.

Maximum Number of Rows

Specifies the maximum number of rows to include per section. If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of sections to include in each stack.

Table Spacing

Specifies the spacing between tables.

Tile Direction

Specifies the direction in which the table tiles (across or down).

## Label New Parts

**NOTE** This property group is displayed when accessing the settings from the CreateNetwork command.

Use these settings to establish the default labeling when you add parts to a network.

Pipe in Plan

Specifies whether a new pipe is labeled by default when placed in plan view.

**Pipe Profile**

Specifies whether a new pipe is labeled by default when placed in profile view.

**Structure in Plan**

Specifies whether a new structure is labeled by default when placed in plan view.

**Structure Profile**

Specifies whether a new structure is labeled by default when placed in profile view.

**Related procedures:**

■ Editing Pipe Network Settings (page 1201)

# Edit Feature Settings - Pipe Dialog Box

Use this dialog box to view and change standard drawing ambient settings for pipe objects.

The standard ambient settings are preceded with ⬜ . For feature-specific pipe object setting information, see Edit Feature Settings - Pipe Network Dialog Box (page 2026).

For more information about this dialog box, see Working with the Standard Settings Dialog Box Controls (page 62).

**Related procedures:**

■ Editing Pipe Network Settings (page 1201)

# Edit Feature Settings - Structure Dialog Box

Use this dialog box to view and change standard drawing ambient settings for structure objects.

The standard ambient settings are preceded with ⬜ . For feature-specific structure object setting information, see Edit Feature Settings - Pipe Network Dialog Box (page 2026).

For more information about this dialog box, see Working with the Standard Settings Dialog Box Controls (page 62).

**Related procedures:**

■ Editing Pipe Network Settings (page 1201)

# Network Layout Tools

Use this toolbar to create or edit a pipe network.

The names of the parts list, surface, and alignment referenced in the currently selected pipe network are displayed at the bottom of this toolbar.



⬛ **Pipe Network Properties**

Opens the Pipe Network Properties (page 2033) dialog box, where you can control defaults and properties of the pipe network, such as the referenced parts list, object references, label defaults, and more.

 Select Surface

Opens the Select Surface dialog box, where you can specify the surface referenced by any new parts to be added to the network. When a surface is selected, it is used to determine depths and elevations of pipe network parts, based on rules for the part. If there are no surfaces in the drawing, this button is not available. Changes made to this selection will not affect parts already created in the pipe network.

 Select Alignment

Opens the Select Alignment dialog box, where you can specify the alignment referenced by any new parts to be added to the network. If there are no alignments in the drawing, this button is not available. Changes made to this selection will not affect parts already created in the pipe network. The alignment provides station offset values for the pipe network parts, including the default alignment for labels.

 Parts List

Opens the Select Parts List dialog box, where you can specify the parts list referenced by the current pipe network. The parts list controls the set of pipes and structures that are available to add in the current pipe network.

Structure List

Specifies the type of structure currently selected for insertion into this pipe network.

Pipe List

Specifies the type of pipe currently selected for insertion into this pipe network.

 Drawing Commands

This drop-down button lets you specify whether to insert both pipes and structures, only pipes, or only structures.

 When Pipes And Structures mode is selected, a structure is inserted at the first selection point, followed by a pipe. Subsequent selection points continue to insert structures connected to pipes.

 When Pipes Only mode is selected, you are prompted to select the start and end points for each pipe you insert. You can only insert pipes in this mode.

 When Structures Only mode is selected, you are prompted to select the insertion points for each structure you insert. You can only insert structures in this mode.

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**NOTE** The initial default command is Pipes and Structures. The initial prompt is `Specify the structure insertion point:` When reopening the toolbar, the default command is the last command that was used.

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 Toggle Upslope/Downslope

Specifies whether the slope of the pipe network is upstream or downstream. This button acts as a toggle.

When  is displayed, the pipe network slope attribute is set to downslope. When is  displayed the slope attribute is set to upslope. This also ensures that pipe rules are processed correctly.

 Delete Pipe Network Object

Deletes the specified pipe network part from the drawing.

 Pipe Network Vistas

Opens the Panorama window (page 102). Lets you display and edit the pipe network data in vistas. There is a vista for pipe object data and a vista for structure object data. For more information, see Pipe Network Vistas (page 2064).

⇦ Undo

Reverses the last action.

# Pipe Network Properties Dialog Box

Use this dialog box to change the pipe network name, description, default settings, layers, labels, and settings for profile and section views.

You can change the style, parts list, surface, alignment, or layers of a selected pipe network. You can also change style and layers for how the pipe network is displayed in profile and section views.

## Information Tab (Pipe Network Properties Dialog Box)

Use this tab to view or change general information for the pipe network.

Name

Specifies the name of the current pipe network.

Description

Specifies the description for the current pipe network.

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Layout Settings Tab (Pipe Network Properties Dialog Box)

Use this tab to view and edit the default parameters used for the current pipe network during layout (plan view) mode.

The parameters include labels, parts list, layers, and name templates.

### Labels

Structure Plan Label Style

Specifies the label style used for structures inserted into this pipe network during layout (plan view) mode.

Pipe Plan Label Style

Specifies the label style used for pipes inserted into this pipe network during layout (plan view) mode.

### Network Parts List

Parts List

Specifies a parts list for this pipe network. Use the parts list to select parts (pipes and structures) that can be inserted into the current pipe network. For more information, see Part Catalog and Parts Lists (page 1237).

### Default Object Reference

Surface Name

Displays the name of the surface that is referenced by this pipe network. The surface provides elevation data to the pipe network objects.

Alignment Name

Displays the name of the alignment that is referenced by this pipe network. The alignment provides station and offset data to the pipe network objects.

### Default Network Layers

Pipe Plan Layer

Specifies the default layer used when creating pipe objects within this pipe network. After the pipe object is created, you can change the layer using any of the standard AutoCAD layer features.

Structure Plan Layer

Specifies the default layer used when creating structure objects within this pipe network. After the structure object is created, you can change the layer using any of the standard AutoCAD layer features.

### Name Templates

Pipes

Specifies the object name template used when creating pipe objects within this pipe network.

Structures

Specifies the object name template used when creating structure objects within this pipe network.

## Profile Tab (Pipe Network Properties)

Use this tab to set default label styles and layers for the pipe network objects when they are displayed in a profile view.

### Labels

Structure Profile Label Style

Specifies the label style used to display the structures within this pipe network in a profile view.

Pipe Profile Label Style

Specifies the label style used to display the pipes within this pipe network in a profile view.

### Default Profile Layers

Pipe Profile Layer

Specifies the default layer used to display the pipes within this pipe network in a profile view.

Structure Profile Layer

Specifies the default layer used to display the structures within this pipe network in a profile view.

## Section Tab (Pipe Network Properties)

Use this tab to set the default layer for the pipe network objects when they are displayed in a section view.

### Default Section Layers

Network Section Layer

Specifies the default layer used to display this pipe network in a section view.

## Statistics Tab (Pipe Network Properties Dialog Box)

Use this tab to display pipe network statistics that are based on the current state of the pipe network.

### General

Displays the general statistics for the current pipe network.

Minimum Elevation

Displays the minimum elevation value found in the pipe network.

Maximum Elevation

Displays the maximum elevation value found in the pipe network.

**Pipes**

Total Number of Pipes

Displays the total number of pipes in this pipe network.

**Structures**

Total Number of Structures

Displays the total number of structures, including null structures, in this pipe network.

Inlet-Outlet Structure

Displays the total number of inlet/outlet structures in this pipe network.

Junction Structure

Displays the total number of junction structures in this pipe network.

Null Structures

Displays the total number of null structures in this pipe network.

**References**

Alignments

Displays the total number of alignments referenced in this pipe network.

Surfaces

Displays the total number of surfaces referenced in this pipe network.

# Network Parts List Dialog Box

Use this dialog box to create a new pipe network parts list, or to view or change the properties and/or contents of a parts list.

# Information Tab (Network Parts List Dialog Box)

Use this tab to view or change general information for the pipe network parts list.

Name

Specifies the name of the current parts list.

Description

Specifies the description for the current parts list.

# Pipes Tab (Network Parts List Dialog Box)

Use this tab to view or change the pipe sizes included in the parts list.

You can add new pipe sizes to the parts list or remove pipe sizes from the parts list. Each size selection matches a part size from a part family in the part catalog. Optional properties may also be set on the part size. The combined set of selected and optional properties is assigned to the pipe when it is inserted into the drawing.

Name

This tree view displays the name of the parts list at the top level, and then the names of the part families included in the parts list, and then the names of the part sizes included in each part family. Note when

a new size is added to the parts list, it is assigned a default unique name. The size name can be edited (renamed) to any unique name within the part family size list.

Style

Specifies the default style assigned to the pipe when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

Rules

Specifies the default rules assigned to the pipe when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

Render Material

Specifies the default render material assigned to the pipe when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

## Structures Tab (Network Parts List Dialog Box)

Use this tab to view or change the structure sizes included in the parts list.

You can add new structure sizes to the parts list or remove structure sizes from the parts list. Each size selection matches a part size from a part family in the part catalog. Optional properties may also be set on the part size. The combined set of selected and optional properties is assigned to the structure when it is inserted into the drawing.

Name

This tree view displays the name of the parts list at the top level, and then the names of the part families included in the parts list, and then the names of the part sizes included in each part family. Note when a new size is added to the parts list, it is assigned a default unique name. The size name can be edited (renamed) to any unique name within the part family size list.

Style

Specifies the default style assigned to the structure when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

Rules

Specifies the default rules assigned to the structure when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

Render Material

Specifies the default render material assigned to the structure when it is inserted into the drawing. Select a part family and click the Select All Edit icon  to assign the selected style to all part sizes within that family.

## Summary Tab (Network Parts List)

Use this tab to view information and statistics related to the part list.

### Information

Displays general information about the current parts list.

### Statistics

Number of Pipes

Displays the total number of pipes in this parts list.

Number of Structures

Displays the total number of structures in this parts list.

# Part Catalog Dialog Box

Use this dialog box to view the contents of the part catalog and to select the items you want to add to a parts list.

The tree view in the left pane displays the available part types in the part catalog. Parts are organized by type (for example, "Catch Basins"), and then by part family ("Concentric Catch Basin 1"). If you access the Part Catalog dialog box from the Pipes tab of the Network Parts List dialog box, only available pipe part families are displayed. Similarly, if you access the Part Catalog dialog box from the Structures tab of the Network Parts List dialog box, only available structure part families are displayed.

When a part family is selected in the left pane, a preview image of the part shape is displayed in the right pane.

**NOTE** Items that are already included in the current parts list are not displayed in the part catalog. Only items that are available to be added to the parts list are displayed in the catalog.

**Related procedures:**

■ Creating a Parts List (page 1238)

# Pipe Rule Set Dialog Box

Use this dialog box to view or edit the set of rules that can be assigned to pipes in a pipe network.

# Information Tab (Pipe Rule Set Dialog Box)

Use this tab to view or edit basic information about this pipe rules set.

You can view or edit the name and optional description values, and you can view information about when the rule set was created and last modified.

# Rules Tab (Pipe Rule Set Dialog Box)

Use this tab to view or edit the rules included in this set of pipe rules.

You can change values of existing rules, add rules, or delete rules. You can also change the order in which rules are processed by changing the order they appear on this dialog. For more information, see Part Rules (page 1247).

**NOTE** Rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Storm Sewers Extension.

**Add Rule**

Click this button to display the Add Rule dialog box (page 2069) where you can select a new rule to add to this rule set.

**Delete Rule**

You can remove a rule from the rule set by selecting the rule on this dialog box and clicking this button.

### Up and Down Arrow Buttons

⇧ ⇩ You can change the order in which rules are processed by changing the order they appear on this dialog. Rules are processed sequentially, starting from the rule displaying at the bottom of this dialog box, and ending with the rule displayed at the top of this dialog box.

### Cover And Slope

This rule ensures that a pipe slopes appropriately and warns when a pipe is placed too close to a ground surface. For more information, see Cover And Slope Rule (page 1251).

Maximum Cover

Specifies the maximum cover of soil over the length of the pipe, based on the surface being referenced by that pipe. If the maximum cover is exceeded, a rule violation occurs. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Maximum Slope

Specifies the maximum slope of the pipe, expressed in percent. If the pipe slope is greater than the maximum, a rule violation occurs for that object.

Minimum Cover

Specifies the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. This is also used to determine the initial elevations of the pipe. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

Minimum Slope

Specifies the minimum slope of the pipe, expressed in percent. During layout, a pipe will be created according to its minimum slope rule value. If the pipe is edited so that its slope is less than the minimum, you can still edit the pipe as desired, breaking the minimum slope rule, but a rule violation occurs for that object.

### Cover Only

This rule is intended for laying out pipes in a pressure-based pipe network where pipe elevations are determined according to a specified depth below a terrain. This rule ensures that the minimum cover is met along the length of the pipe, and also validates that both the minimum and maximum cover values are not violated along any length of the pipe. For more information, see Cover Only Rule (page 1252).

Maximum Cover

This is the maximum cover of soil over the pipe, based on the surface being referenced by that pipe. If the pipe cover exceeds the maximum cover value, a rule violation occurs for that object. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Minimum Cover

This rule represents the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

### Length Check

This rule governs the behavior that produces a warning condition on a pipe if the pipe length exceeds the value specified for the maximum pipe length, or is less than the value specified for the minimum pipe length. For more information, see Length Check Rule (page 1252).

Maximum Length

Specifies a maximum length for pipes. When this rule is in use, you can still draw pipes that exceed the defined maximum pipe length. However, the object will be displayed with a warning icon in the Prospector

list view. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Minimum Length

Specifies a minimum length for pipes. When this rule is in use, you can still draw pipes that are shorter than the defined minimum pipe length. However, the object will be displayed with a warning icon in the Prospector list view. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is not met.

### Pipe To Pipe Match

This rule governs how a pipe elevation is determined in a pipe network that contains only pipes (no structures), and or when a pipe is inserted onto an existing pipe to break the pipe. For more information, see Pipe To Pipe Match Rule (page 1253).

Match Location

This parameter controls whether the inserted pipe holds to the pipe's invert, crown, or centerline elevation (location).

Drop Value

The drop value on a structure can determine a pipe's depth.

# Pipe Properties Dialog Box

Use this dialog box to view or change the properties of a pipe object.

You can change the name of the pipe object, description, style, render material, and more. You can also change part properties and rules.

When you add a pipe to a pipe network, much of the information about the pipe, such as its description and properties, is derived from the definition of the pipe in the part catalog. After the pipe is inserted into a drawing, you can make changes using this dialog box. When you make changes to the pipe, these changes apply only to the selected pipe in your drawing. The pipe definition in the part catalog is not affected by your modifications in the drawing.

For descriptions of part properties, see Part Properties (page 1235).

# Information Tab (Pipe Properties Dialog Box)

Use this tab to change the name, description, and style information for the pipe.

Name

Specifies the name of the current pipe.

Description

Specifies an optional description of the current pipe.

Object Styles

Specifies the pipe style. Select other styles from the list.

Style

Specifies the style options. Click to create a new style, copy or edit the current style selection, or pick a style from drawing. Click to open the Style Detail dialog box, where you can preview the style and creation information

Render Material

Specifies the default render material assigned to the pipe when it is inserted into the drawing. Click ![icon]
to select a render material from the drawing.

Show Tooltips

Controls whether or not tooltips are displayed for the object in the drawing.

# Part Properties Tab (Pipe Properties Dialog Box)

Use this tab to view or edit properties associated with this pipe.

This tab displays the properties associated with this pipe, along with their current values. Pipe properties
define a variety of characteristics of the pipe object, including its basic size and shape, and certain behaviors
such as flow direction and conditions under which the pipe part will automatically adjust its size.

Certain properties are specific to a type of pipe. For example, circular shaped pipes use the Inner Pipe Diameter
property while rectangular shaped pipes use the Inner Pipe Width and Inner Pipe Height properties.

Some properties are read-only and others are editable. To change an editable property, double-click on the
Value field and enter in a new value, or select one from the list. For more information, see Part Properties
(page 1235).

## General

These properties specify general characteristics of the pipe, such as the flow direction method, flow direction,
and the surface and alignment referenced.

Pipe Flow Direction Method

Specifies the method that is used to determine the flow for this pipe. Option are bi-directional, start to
end, end to start, or according to the current slope of the pipe.

Flow Direction

Specifies the current flow direction of the pipe, relative to the start and endpoint of the pipe. When you
begin drawing a pipe, the flow direction is based on the direction in which you draw the pipe.

Reference Surface

Specifies the surface of the pipe.

Reference Alignment

Specifies the alignment of the pipe.

## Geometry

These properties specify characteristics such as the names of the start and end structures, bearing, station
data, offsets, slope data, elevation data, and more.

Pipe Start Structure

Specifies the name of the structure connected to the start of the pipe.

Pipe End Structure

Specifies the name of the structure connected to the end of the pipe.

Bearing

Specifies pipe's bearing.

Pipe Start Station

Specifies pipe's start station.

Pipe End Station

Specifies pipe's end station.

**Start Offset**

Specifies the offset for the start of the pipe.

**End Offset**

Specifies the offset for the end of the pipe.

**Pipe Slope (Hold Start)**

Specifies the slope of the pipe away from the pipe's start point.

**Pipe Slope (Hold End)**

Specifies the slope of the pipe away from the pipe's end point.

**Pipe Slope**

Specifies the pipe's slope in absolute value.

**Start Invert Elevation**

Specifies the pipe's starting point elevation according to the pipe's invert.

**End Invert Elevation**

Specifies the pipe's ending point elevation according to the pipe's invert.

**Start Crown Elevation**

Specifies the pipe's starting point elevation according to the pipe's crown.

**End Crown Elevation**

Specifies the pipe's ending point elevation according to the pipe's crown.

**Pipe Start Easting**

Specifies the easting of the pipe's starting point.

**Pipe Start Northing**

Specifies the northing of the pipe's starting point.

**Pipe End Easting**

Specifies the easting of the pipe's ending point.

**Pipe End Northing**

Specifies the northing of the pipe's ending point.

**Start Centerline Elevation**

Specifies the pipe's starting point elevation according to the pipe's centerline.

**End Centerline Elevation**

Specifies the pipe's ending point elevation according to the pipe's centerline.

**Minimum Cover**

Specifies the minimum depth of cover along the entire length of pipe, from the top outside of the pipe to the reference surface.

**Maximum Cover**

Specifies the maximum depth of cover along the entire length of pipe, from the top outside of the pipe to the reference surface.

**2D Length - Center To Center**

Specifies the two-dimensional length of the pipe, measured from the center of the connected starting structure to the center of the connected ending structure.

**3D Length - Center To Center**

Specifies the three-dimensional length of the pipe, measured from the center of the connected starting structure to the center of the connected ending structure.

**2D Length - To Inside Edges**

Specifies the two-dimensional length of the pipe, measured from the inside edge of the connected starting structure to the inside edge of the connected ending structure.

**3D Length - To Inside Edges**

Specifies the three-dimensional length of the pipe, measured from the inside edge of the connected starting structure to the inside edge of the connected ending structure.

## Resize Behavior

This property specifies the pipe behavior that occurs when the pipe is automatically resized.

**On Resize, Hold**

Specifies if the pipe will hold its invert, crown, or centerline elevation when the pipe is automatically resized due to an edit.

## Part Data

These properties specify a variety of part characteristics, such as part type, part subtype, and the part size name as defined in the part catalog. Some of these properties specify the dimensions that define the part's basic shape and therefore are read-only.

**Part Type**

Specifies pipe's part type.

**Part Subtype**

Specifies pipe's subtype.

**Part Description**

Specifies pipe's description.

**Part Size Name**

Specifies the part size name.

**Cross Sectional Shape**

Specifies the pipe's cross sectional shape, such as circular, egg-shaped, elliptical, or rectangular.

**Wall Thickness**

Specifies the wall thickness for this pipe, measured from the inside edge of the pipe to the outside edge of the pipe.

**Material**

Specifies the material defined for this part. This optional property is used mainly for labeling purposes. It allows you to identify the type of material used to construct this part.

**Minimum Curve Radius**

Specifies the minimum curve radius for this part.

**Manning Coefficient**

This optional property specifies a Manning Coefficient value for a given part size.

**Hazen Williams Coefficient**

This optional property specifies a Hazen Williams Coefficient value for a given part size.

**Darcy Weisbach Factor**

This optional property specifies a Darcy Weisbach Factor value for a given part size.

**Inner Pipe Diameter**

Specifies the inner diameter for this pipe. This property is used for circular shaped pipes only.

Inner Pipe Width

Specifies the inner width for this pipe. This property is used for non-circular shaped pipes only, such as rectangular and egg-shaped.

Inner Pipe Height

Specifies the inner height for this pipe. This property is used for non-circular shaped pipes only, such as rectangular and egg-shaped.

### Hydraulic Properties

These properties specify the hydraulic property information related to the pipe. They are responsible for displaying the hydraulic grade line and energy grade line. Hydraulic grade lines and energy grade lines are displayed as a straight line through structures, connected between the HGL Up or EGL Up property from one pipe to the HGL Down or EGL Down property of the next pipe. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280). For information about creating label styles for displaying labels for these properties, see Labeling Hydraulic Properties in Pipe Networks (page 1270).

Hydraulic Grade Line Up

Specifies the elevation of the hydraulic grade line for pipe networks flowing in an upstream direction that contain hydraulic property data.

Hydraulic Grade Line Down

Specifies the elevation of the energy grade line for pipe networks flowing in a downstream direction that contain hydraulic property data.

Energy Grade Line Up

Specifies the elevation of the energy grade line for pipe networks flowing in an upstream direction that contain hydraulic property data.

Energy Grade Line Down

Specifies the elevation of the energy grade line for pipe networks flowing in a downstream direction that contain hydraulic property data.

Flow Rate

Specifies the flow rate for pipe networks that contain hydraulic property data.

Junction Loss

Specifies the junction loss for pipe networks that contain hydraulic property data.

Return Period

Specifies the return period for pipe networks that contain hydraulic property data.

## Rules Tab (Pipe Properties Dialog Box)

Use this tab to view or change the rule set or rule values associated with this pipe.

For more information, see Pipe Rules (page 1251) and Editing Part Rules (page 1258).

### Rule Set

This drop-down list lets you specify the rule set for the pipe.

### Use Values From Rule Set

This option lets you choose to override or accept the rule values as they are defined in the specified rule set. If you clear this check box, you can edit the rule values on this dialog box, and the pipe uses the rule values as they are specified on this dialog box. If this option is selected, the pipe uses the rule values as they are defined in the specified rule set, and those values display on this dialog box.

## Cover And Slope

This rule ensures that a pipe slopes appropriately and warns when a pipe is placed too close to a ground surface. For more information, see Cover And Slope Rule (page 1251).

Maximum Cover

> Specifies the maximum cover of soil over the length of the pipe, based on the surface being referenced by that pipe. If the maximum cover is exceeded, a rule violation occurs. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Maximum Slope

> Specifies the maximum slope of the pipe, expressed in percent. If the pipe slope is greater than the maximum, a rule violation occurs for that object.

Minimum Cover

> Specifies the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. This is also used to determine the initial elevations of the pipe. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

Minimum Slope

> Specifies the minimum slope of the pipe, expressed in percent. During layout, a pipe will be created according to its minimum slope rule value. If the pipe is edited so that its slope is less than the minimum, you can still edit the pipe as desired, breaking the minimum slope rule, but a rule violation occurs for that object.

## Cover Only

This rule is intended for laying out pipes in a pressure-based pipe network where pipe elevations are determined according to a specified depth below a terrain. This rule ensures that the minimum cover is met along the length of the pipe, and also validates that both the minimum and maximum cover values are not violated along any length of the pipe. For more information, see Cover Only Rule (page 1252).

Maximum Pipe Cover

> This is the maximum cover of soil over the pipe, based on the surface being referenced by that pipe. If the pipe cover exceeds the maximum cover value, a rule violation occurs for that object. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Minimum Pipe Cover

> This rule represents the minimum cover of soil over the pipe, based on the surface being referenced by that pipe. During layout, a pipe will be created that attempts to maintain the minimum cover. If the pipe is edited so that its cover is less than the minimum cover value, a rule violation occurs for that object.

## Length Check

This rule governs the behavior that produces a warning condition on a pipe if the pipe length exceeds the value specified for the maximum pipe length, or is less than the value specified for the minimum pipe length. For more information, see Length Check Rule (page 1252).

Maximum Pipe Length

> Specifies a maximum length for pipes. When this rule is in use, you can still draw pipes that exceed the defined maximum pipe length. However, the object will be displayed with a warning icon in the Prospector list view. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

Minimum Pipe Length

> Specifies a minimum length for pipes. When this rule is in use, you can still draw pipes that are shorter than the defined minimum pipe length. However, the object will be displayed with a warning icon in the

Prospector list view. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is not met.

**Pipe To Pipe Match**

This rule governs how a pipe elevation is determined in a pipe network that contains only pipes (no structures), and or when a pipe is inserted onto an existing pipe to break the pipe. For more information, see Pipe To Pipe Match Rule (page 1253).

Match Location

This parameter controls whether the inserted pipe holds to the pipe's invert, crown, or centerline elevation (location).

Drop Value

The drop value on a structure can determine a pipe's depth.

# Pipe Style Dialog Box

Use this dialog box to control the display of various pipe components in a pipe network.

You can edit styles to control how components like pipes walls, pipe ends, pipe centerlines, hatchings and crossings display in plan, profile, and section views.

Create different styles to use in the various phases of your project. For example, you can create a style to use specifically in the design phase, displaying pipe components in different colors, and create another style to use for plotting, displaying the components differently.

# Information Tab (Pipe Style Dialog Box)

Use this tab to change the pipe style name and description information, and to review details about the pipe style, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Plan Tab (Pipe Style Dialog Box)

Use this tab to define the appearance of the pipe in a 2D plan view.

**Pipe Wall Sizes**

Specifies the method that will be used to define and draw the dimensions of the pipe wall.

Use Part Dimensions

The inner and outer pipe wall dimensions are drawn according to the actual pipe dimensions.

User Defined

The inner and outer pipe wall dimensions are drawn according to user-specified values. The following size options are available:

■ Use Drawing Scale: When this option is selected, the value specified in the Units edit box (inches or millimeters, when the drawing linear unit is set to feet or meters, respectively) will be multiplied by the drawing scale to determine the width of the pipe walls displayed in the drawing. The user then enters the desired inner and outer wall dimensions in the text boxes below.

■ Use Size As Percentage Of Screen: When this option is selected, the value specified in the Units edit box is used as a percentage of the drawing screen size. The width of the pipe walls displayed in the drawing will be the displayed at the same percentage as the drawing screen size specified.

■ Use Size In Absolute Units: When this option is selected, the specified value that is entered in the Units edit box is an absolute value in the drawing linear units. The title text above this box displays Feet when the drawing units are set to feet, and Meters when the drawing units are set to meters.

## Pipe Hatch Options

Specifies which components of the pipe shape display a hatch pattern.

Hatch To Inner Walls

The hatch pattern is displayed only in the central area of the pipe, and to not hatch the pipe wall. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is not hatched.

Hatch To Outer Walls

The hatch pattern is displayed to the entire area of the pipe, including the pipe wall. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is hatched.

Hatch Walls Only

The hatch pattern is displayed only in the pipe walls. The pipe wall is the area between the pipe inner wall and the pipe outer wall.

## Pipe End Line Size

Specifies the method that will be used to define and draw the dimensions of the pipe ends.

Draw To Inner Walls

The pipe end line draws to the inner walls of the pipe.

Draw To Outer Walls

The pipe end line draws to the outer walls of the pipe.

User Defined

The pipe end line draws to a size specified by the user. The size options available are the same as the options available for the pipe wall dimensions.

## Pipe Centerline Options

Specifies the method that will be used to define and draw the dimensions of the pipe centerline.

By Lineweight

The pipe centerline is drawn according to the currently specified line weight of the pipe.

Specify Width

The pipe centerline is drawn according to user-specified values. The following width options are available:

■ Draw To Inner Walls: When this option is selected, the value specified in the edit box below (inches or millimeters, when the drawing linear unit is set to feet or meters, respectively) will be multiplied by the drawing scale to determine the width of the pipe centerline displayed in the drawing. The pipe centerline draws to the inner walls of the pipe, according to the value specified in the text box below.

■ Draw To Outer Walls: When this option is selected, the value specified in the edit box below (inches or millimeters, when the drawing linear unit is set to feet or meters, respectively) will be multiplied by the drawing scale to determine the width of the pipe centerline displayed in the drawing. The pipe centerline draws to the outer walls of the pipe, according to the value specified in the text box below.

■ Use Drawing Scale: When this option is selected, the value specified in the Units edit box (inches or millimeters, when the drawing linear unit is set to feet or meters, respectively) will be multiplied by the drawing scale to determine the width of the pipe walls displayed in the drawing. The user then enters the desired inner and outer wall dimensions in the text boxes below.

- Use Size As Percentage Of Screen: When this option is selected, the value specified in the Units edit box is used as a percentage of the drawing screen size. The width of the pipe walls displayed in the drawing will be the displayed at the same percentage as the drawing screen size specified.

- Use Size In Absolute Units: When this option is selected, the specified value that is entered in the Units edit box is an absolute value in the drawing linear units. The title text above this box displays Feet when the drawing units are set to feet, and Meters when the drawing units are set to meters

### Align Hatch To Pipe

When this option is selected, the angle of the hatch lines for the pipe align with the angle of the pipe.

### Clean Up Pipe To Pipe Connections

When this option is selected, the appearance of pipe to pipe connections in plan view is cleaned up. This option is available for multiple adjoining pipes that are connected to other pipes with null structures. This option must be selected for the clean up to occur, and the cleanup occurs only on the pipes that are connected to other pipes with null structures between them. For more information, see Pipe End Cleanup Option (page 1223). You can also set this option on the Profile tab of the Pipe Style dialog box for pipe to pipe connections displayed in profile views.

## Profile Tab (Pipe Style Dialog Box)

Use this tab to define the appearance of the pipe in a profile view.

### Pipe Wall Sizes

Specifies the method that will be used to define and draw the dimensions of the pipe wall.

Use Part Dimensions

The inner and outer pipe wall dimensions are drawn according to the actual pipe dimensions.

User Defined

The inner and outer pipe wall dimensions are drawn according to user-specified values. The following size options are available:

- Use Drawing Scale: Specifies that the value in the Units edit box (inches, or millimeters, when the drawing linear unit is set to Feet or Meters, respectively) will be multiplied by the drawing scale to determine the width of the pipe walls displayed in the drawing. Then the user enters the desired inner and outer wall dimensions in the text boxes below.

- Use Size As Percentage Of Screen: Specifies that the value in the Units edit box is a percentage of the drawing screen size. The width of the pipe walls displayed in the drawing will always be the percentage specified of the drawing screen size.

- Use Size In Absolute Units: Specifies that the value entered in the Units edit box is an absolute value of the drawing linear units. The static text of the Units edit box displays Feet when the drawing units are set to Feet, and Meters when the drawing units are set to Meters.

### Pipe Hatch Options

Specifies which components of the pipe shape display a hatch pattern.

Hatch To Inner Walls

The hatch pattern is displayed only in the central area of the pipe, and not in the pipe wall area of the pipe. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is not hatched.

Hatch To Outer Walls

The hatch pattern is displayed in the entire area of the pipe, including the pipe wall area. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is hatched.

Hatch Walls Only

The hatch pattern is displayed only in the pipe wall area. The pipe wall area is the area between the pipe inner wall and the pipe outer wall.

## Pipe End Line Size

Specifies the method that will be used to define and draw the dimensions of the pipe ends.

Draw To Inner Walls

The pipe end line draws to the inner walls of the pipe.

Draw To Outer Walls

The pipe end line draws to the outer walls of the pipe.

User Defined

The pipe end lines draw to a size specified by the user. The size options available are the same as the options available for the pipe wall dimensions.

## Crossing Pipe Hatch Options

Specifies which components of a pipe crossing display a hatch pattern.

Hatch To Inner Walls

The hatch pattern is displayed only to the inner walls of pipe crossings.

Hatch To Outer Walls

The hatch pattern is displayed all the way to the outer walls of pipe crossings.

Hatch Walls Only

The hatch pattern is displayed only within the pipe wall area of pipe crossings.

## Align Hatch To Pipe

When this option is selected, the angle of the hatch lines for the pipe align with the angle of the pipe.

## Clean Up Pipe To Pipe Connectons

When this option is selected, the appearance of pipe to pipe connections in profile view is cleaned up. This option is available for multiple adjoining pipes that are connected to other pipes with null structures. This option must be selected for the clean up to occur, and the cleanup occurs only on the pipes that are connected to other pipes with null structures between them. For more information, see Pipe End Cleanup Option (page 1223). You can also set this option on the Plan tab of the Pipe Style dialog box for pipe to pipe connections displayed in plan views. By default, this option is not checked.

# Section Tab (Pipe Style Dialog Box)

Use this tab to define the appearance of the pipe in a section view.

## Crossing Pipe Hatch Options

Specifies which components of a pipe crossing display a hatch pattern.

Hatch To Inner Walls

The hatch pattern is displayed only in the central area of the pipe, and does not hatch the pipe wall. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is not hatched.

Hatch To Outer Walls

The hatch pattern is displayed the entire area of the pipe, including the pipe wall. When this option is selected, the pipe wall (which is the area between the inner wall and the outer wall of the pipe) is hatched.

Hatch Walls Only

The hatch pattern is displayed only in the pipe walls. The pipe wall is the area between the pipe inner wall and the pipe outer wall.

# Display Tab (Pipe Style Dialog Box)

Use this tab to change the display and visibility of pipe object components.

## View Direction

Some object styles can have unique display values and varying numbers of displayed components, depending on whether they are being displayed in Plan, Profile, Section, or Model views. You can create and set style characteristics for each supported view direction type using the View Direction list.

Plan

Specifies the display style settings when the pipe is displayed in plan view.

Model

Specifies the display style settings when the pipe is displayed in model view.

Profile

Specifies the display style settings when the pipe is displayed in a profile view.

Section

Specifies the display style settings when the pipe is displayed in a section view.

## Component Display

You can set the following display style characteristics for pipe components: Visibility, Layer, Color, Linetype, LT Scale, and Lineweight.

Pipe Centerline

The style for pipe centerlines.

Inside Pipe Walls

The style for the inside walls of pipes.

Outside Pipe Walls

The style for the outside walls of pipes.

Pipe End Line

The style for the ends of pipes.

Pipe Hatch

The hatch style for pipes.

Pipe Solid

The style for pipes displayed in 3D model view.

Crossing Pipe Inside Walls

The style for the inside wall of pipe crossings displayed in profile or section view.

Crossing Pipe Outside Walls

The style for the outside wall of pipe crossings displayed in profile or section view.

Crossing Pipe Hatch

The hatch style for pipe crossings displayed in profile or section view.

Hydraulic Grade Line

The style for displaying hydraulic grade lines for pipe networks that are displayed in profile views. For more information, see Viewing Hydraulic Data in Profile Views (page 1282).

Energy Grade Line

The style for displaying energy grade lines for pipe networks that are displayed in profile views. For more information, see Viewing Hydraulic Data in Profile Views (page 1282).

In certain views, you can specify hatch display styles for the fill areas of the following pipe components:

**Pipe Hatch**: The hatch pattern for pipe components displayed in plan or profile view.

**Crossing Pipe Hatch**: The hatch pattern for pipe crossings displayed in profile or section view.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Pipe Style Dialog Box)

Use this tab to review all the information about the current pipe style.

This information can be copied and pasted to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Structure Rule Set Dialog Box

Use this tab to view or edit basic information about this structure rules set.

You can view or edit the name and optional description values, and you can view information about when the rule set was created and last modified.

## Information Tab (Structure Rule Set Dialog Box)

Use this tab to view or edit the basic information about this structure rules set, such as a name and an optional description for this set of structure rules.

## Rules Tab (Structure Rule Set Dialog Box)

Use this tab to view or edit the rules included in this set of structure rules.

You can change values of existing rules, add rules, and or delete rules. You can also change the order in which rules are processed by changing the order in which they appear on this dialog. For more information, see Part Rules (page 1247).

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**NOTE** Rules are not applied to pipes or structures when importing pipe networks from either LandXML or from the Storm Sewers Extension.

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**Add Rule**

Click this button to display the Add Rule dialog box (page 2069) where you can select a new rule to add to this rule set.

**Delete Rule**

You can remove a rule from the rule set by selecting the rule on this dialog box and clicking this button.

**Up and Down Arrow Buttons**

⇧ ⇩ You can change the order in which rules are processed by changing the order they appear on this dialog. Rules are processed sequentially, starting from the rule displaying at the bottom of this dialog box, and ending with the rule displayed at the top of this dialog box.

**Pipe Drop Across Structure**

This rule compares all pipes connected to a single structure and ensures that pipes enter and exit the structure at logical locations. For more information, see Pipe Drop Across Structure Rule (page 1253).

Drop Reference Location

  Determines the drop location by using the pipe's invert, crown, or centerline elevation.

Drop Value

  Specifies what the drop value is between the lowest incoming pipe and any outgoing pipe connected to the structure.

Maximum Drop Value

  Specifies the maximum drop value between the lowest incoming pipe and any outgoing pipe connected to the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

**Maximum Pipe Size Check**

This rule checks to see if pipes entering a structure have a diameter or width that exceeds a specified maximum value. For more information, see Maximum Pipe Size Check Rule (page 1257).

Maximum Pipe Diameter or Width

  For circular pipes, this parameter measures the pipe diameter. For rectangular pipes, it measures width. If a pipe diameter or width exceeds the maximum value, a warning is issued on the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

**Set Sump Depth**

This rule specifies the sump depth, or the vertical distance from the invert of the lowest pipe attached to the structure to the inside bottom of the Structure, see Set Sump Depth Rule (page 1258).

Sump Depth

  Determines the sump depth.

Drop Value

  Specifies what the drop value is between the lowest incoming pipe and any outgoing pipe connected to the structure.

Maximum Drop Value

  Specifies the maximum drop value between the lowest incoming pipe and any outgoing pipe connected to the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

# Structure Properties Dialog Box

Use this dialog box to view or change the properties of a structure object.

You can change the name of the structure object, description, style, render material, and more. You can also change part properties and rules.

When you add pipe network containing structures to a drawing, much of the information about the structure, such as its description and properties, is derived from the definition of the structure as it exists in the part catalog.

After a structure is inserted into a drawing, you can make changes to the structure using this dialog box, if desired. When you make changes using this dialog box, the changes apply only to the current structure selected in your drawing, and have no impact on the information defined for that structure in the part catalog. For descriptions of part properties, see Part Properties (page 1235).

## Information Tab (Structure Properties Dialog Box)

Use this tab to change the name, description, and style information for the structure.

Name
  Specifies the name of the current pipe.

Description
  Specifies an optional description of the current pipe.

Object Styles
  Specifies the pipe style. Select other styles from the list.

Style
  Specifies the style options. Click to create a new style, copy or edit the current style selection, or pick a style from drawing. Click  to open the Style Detail dialog box, where you can preview the style and creation information.

Render Material
  Specifies the default render material assigned to the pipe when it is inserted into the drawing. Click  to select a render material from the drawing.

Show Tooltips
  Controls whether or not tooltips are displayed for the object in the drawing.

## Part Properties Tab (Structure Properties Dialog Box)

Use this tab to view or change properties associated with this structure.

This tab displays the properties associated with this structure, along with their current values. Structure properties define a variety of characteristics of the structure object, including its basic size and shape and elevation adjustment behavior.

Some properties are specific to a certain type of structure. For example, only junction structures use rim and sump behavior properties. Only inlet-outlet structures use properties that define headwalls such as Headwall Base Width.

Some properties are read-only and others are editable. To change an editable property, double-click on the Value field and enter a new value, or select one from the list. For more information, see Part Properties (page 1235)

### General

These properties specify general characteristics of the structure, such as the structure elevation at its insertion point, and the surface and alignment referenced.

Surface Elevation At Insertion Point
  Specifies the elevation at the structure's insertion point.

Reference Surface

Specifies the surface referenced by the structure.

Reference Alignment

Specifies the alignment referenced by the structure.

## Geometry

These properties specify characteristics such as the structure's rotation angle, offset, station data, and the number of pipes that are currently connected to the structure.

Structure Rotation Angle

Specifies the structure's rotation angle.

Structure Offset

Specifies the structure's offset.

Structure Northing

Specifies the structure's northing.

Structure Easting

Specifies the structure's easting.

Connected Pipes

Specifies the number of pipes currently connected to the structure.

## Insertion Rim Behavior

These properties are associated with junction structures only. They specify the structure's rim elevation and adjustment behavior.

Insertion Rim Elevation

Specifies the current elevation of the structure rim. This cannot be edited if Automatic Surface Adjustment property is set to True.

Automatic Surface Adjustment

Specifies whether or not to automatically adjust the rim elevation to the referenced surface, including the Surface Adjustment Value. When this property is set to True, grip editing the structure modifies the surface adjustment factor. When this property is set t o False, grip editing the structure modifies the rim elevation.

Surface Adjustment Value

If Automatic Surface Adjustment is set to True, the rim elevation is automatically adjusted by the surface elevation, plus the value specified here. If the structure is moved, resulting in the structure elevation changing, the structure automatically adjusts its elevation.

## Sump Behavior

These properties are associated with junction structures only. They specify the structure's behavior, elevation, and the depth of the structure's sump.

Sump Elevation

Specifies the elevation of the structure's sump, measured at the inside bottom of the structure. When this property is enabled, grip editing the structure modifies the sump elevation.

Sump Depth

Specifies the depth of the sump area, measured from the inside bottom of the structure to the lowest pipe invert of all pipes connected to the structure. When this property is enabled, grip editing the structure modifies the sump depth.

Control Sump By

Specifies if the sump depth is controlled by the depth based on the lowest pipe invert, or by a specific elevation value of the sump.

**Part Data**

These properties specify a variety of part characteristics, such as part type, part subtype, and the part size name as defined in the part catalog. Some of these properties define the part's basic shape and behavior, and therefore are read-only.

Part Type

Specifies structure's part type.

Part Subtype

Specifies structure's subtype.

Part Description

Specifies structure's description.

Part Size Name

Specifies the part size name.

Structure Shape

Specifies the basic primitive shape of the structure, as defined from the part catalog. For example, headwalls are typically defined in the part catalog as Bounding box shape, and manholes are typically defined as cylindrical shaped.

Vertical Pipe Clearance

Specifies the clearance required from the top outside of the highest pipe connected to the structure, to the rim. This is defined in the catalog and ensure that pipes enter the structure at an appropriate elevation. For example, it prevents a pipe from entering through the cone of structure.

Rim To Sump Height

Specifies the distance between the sump to the structure's rim. This property is for junction structures only.

Headwall Base Width

Specifies the width of the base of the headwall. This property is for inlet-outlet structures only.

Headwall Base Thickness

Specifies the thickness of the base of the headwall. This property is for inlet-outlet structures only.

Wall Thickness

Specifies the wall thickness for this structure, measured from the inside edge of the structure to the outside edge of the structure. This property is for junction structures only.

Floor Thickness

Specifies the thickness of the bottom of the structure. This property is for junction structures only.

Material

Specifies the material defined for this part. This optional property is used mainly for labeling purposes. It allows you to identify the type of material used to construct this part.

Frame

This optional property specifies the model or type of frame used for a structure.

Grate

This optional property specifies the model or type of grate used for a structure intended to be used as a catchbasin.

Cover

This optional property specifies the model or type of cover used for a structure intended to be used as a manhole.

Frame Height

Specifies the height of the structure's frame. This property is for junction structures only.

Frame Diameter

Specifies the diameter of the structure's frame. This property is for junction structures only.

Frame Length

Specifies the length of the structure's frame. This property is for junction structures only.

Frame Width

Specifies the width of the structure's frame. This property is for junction structures only.

Barrel Height

Specifies the height of the barrel section of the structure. This property is for two-tiered junction structures only.

Barrel Pipe Clearance

This property lets you determine how close to the structure's rim you can position a pipe. It is used for two-tiered junction structures only and specifies the clearance required above the top of the barrel section of the structure. It is similar to Vertical Pipe Clearance, but only describes the transition zone between the access cylinder and the larger barrel cylinder of a two-tiered structure.

Cone Height

Specifies the height of the cone shaped section of the structure. This property is for two-tiered junction structures only.

Slab Thickness

Specifies the thickness of the structure slab. This property is for junction structures only.

Inner Structure Diameter

Specifies the inner diameter for this structure. This property is for junction structures only.

Structure Height

Specifies the total height for this structure, measured from the bottom outside edge of the structure to the top outside edge of the structure.

Structure Diameter

Specifies the diameter for this structure, measured from the outside edges of the structure. Circular shaped structures typically use the diameter property while non-circular structures use width and length.

Structure Width

Specifies the width for this structure, measured from the outside edges of the structure. Non-circular shaped structures typically use the width and length properties while circular structures use diameter.

Structure Length

Specifies the length for this structure, measured from the outside edges of the structure. Non-circular shaped structures typically use the width and length properties while circular structures use diameter.

## Hydraulic Properties

These properties specify the hydraulic grade line, the energy grade line, and the junction loss values for pipe networks that contain this data.

Hydraulic grade lines and energy grade lines, when present, are displayed as a straight line through structures, connected between the HGL Up or EGL Up property from one pipe to the HGL Down or EGL Down property

of the next pipe. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

Hydraulic Grade Line
   Specifies the elevation of the hydraulic grade line for pipe networks that contain this data.

Energy Grade Line
   Specifies the elevation of the energy grade line for pipe networks that contain this data.

Known Capacity
   Specifies the known capacity for pipe networks that contain this data.

Bypass Target
   Specifies the bypass target for pipe networks that contain this data.

## Connected Pipes Tab (Structure Properties Dialog Box)

Use this tab to quickly make adjustments to pipes that are connected to a single structure.

The Connected Pipes tab lets you quickly view and edit properties that are important for pipes connected to structures. For example, you can use this tab to edit the following properties on pipes connected to the selected structure:

■ inner diameter

■ inner width

■ invert elevation

■ centerline elevation

■ crown elevation

You can use this tab to quickly perform a variety of edits to pipes that are connected to a structure. You can align the elevations on multiple pipes connected to the structure according to the invert, crown, or centerline elevation of a selected pipe.

For more information, see Matching Elevations on Connected Pipes (page 1219). The following information describes the properties that are displayed on the Connected Pipes tab.

Name
   Displays the name of the current structure and each pipe connected to it. Read-only.

Status
   Displays the current status of each pipe connected to the structure. Read-only.

Description
   Displays the description of each pipe connected to the structure. Read-only.

Inner Diameter
   Specifies the inner diameter for circular pipes. Editable.

Inner Width
   Specifies the inner width for non-circular (rectangular) pipes. Editable.

Inner Height
   Specifies the inner height for non-circular (rectangular) pipes. Editable.

Invert Elevation
   Specifies the invert elevation of the pipe. Editable.

Centerline Elevation
Specifies the centerline elevation of the pipe. Editable.

Crown Elevation
Specifies the crown elevation of the pipe. Editable.

Slope
Specifies the slope of the pipe coming out of the structure. Read-only.

In/Out
Displays the flow direction of the pipe, indicating if the pipe flows in to or out of the structure. Read-only.

From Structure
Displays the name of the structure at the other end of the pipe. Read-only.

# Rules Tab (Structure Properties Dialog Box)

Use this tab to view or change the rule set or rule values associated with this structure.

For more information, see Structure Rules (page 1253) and Editing Part Rules (page 1258).

## Rule Set

This drop-down list lets you specify the rule set for the structure.

## Use Values From Rule Set

This option lets you choose to override or accept the rule values as they are defined in the specified rule set. If you clear this check box, you can edit the rule values on this dialog box, and the structure uses the rule values as they are specified on this dialog box. If this option is selected, the structure uses the rule values as they are defined in the specified rule set, and those values display on this dialog box.

## Pipe Drop Across Structure

This rule compares all pipes connected to a single structure and ensures that pipes enter and exit the structure at logical locations. For more information, see Pipe Drop Across Structure Rule (page 1253).

Drop Reference Location
Determines the drop location by using the pipe's invert, crown, or centerline elevation.

Drop Value
Specifies what the drop value is between the lowest incoming pipe and any outgoing pipe connected to the structure.

Maximum Drop Value
Specifies the maximum drop value between the lowest incoming pipe and any outgoing pipe connected to the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

## Maximum Pipe Size Check

This rules checks to see if pipes entering a structure have a diameter or width that exceeds a specified maximum value. For more information, see Maximum Pipe Size Check Rule (page 1257).

Maximum Pipe Diameter or Width
For circular pipes, this parameter measures the pipe diameter. For rectangular pipes, it measures width. If a pipe diameter or width exceeds the maximum value, a warning is issued on the structure. Note that this parameter provides validation only; it does not alter (move or resize) the part in the drawing in any way. It simply produces a rule violation on the part if the specified value is exceeded.

# Structure Style Dialog Box

Use this dialog box to control the display of each structure component in a pipe network.

You can edit styles to control how structure components like boundaries and hatchings display in model, plan, profile, and section views.

Create different styles to represent different types of structures. You can also create different styles to use in the various phases of your project. For example, you can create a style to use specifically in the design phase, displaying pipe network structures in different colors, and create another style to use for plotting, displaying the components differently.

## Information Tab (Structure Style Dialog Box)

Use this tab to change the structure style name and description information, and to review details about the structure style, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

## Model Tab (Structure Style Dialog Box)

Use this tab to define the appearance of the 3D object used to represent the structure in model view.

### Structure

Specifies the method that will be used to define and draw the dimensions of the structure in model view.

Use Catalog Defined 3D Part

The 3D object defined in the parts catalog is used when displaying the structure in 3D. For more information, see Part Catalog and Parts Lists (page 1237). This option is enabled by default.

Use Simple 3D Part

The structure is displayed with one of the following simple shape options:

- Cylinder: The cylinder shape is the default. The dimensions of the cylinder are determined by the boundary of the part. The preview image shows an elongated cylinder in isometric view.

- Cube: The dimensions of the cube are determined by the boundary of the part. The preview image shows an elongated cube in isometric view.

- Sphere: The dimensions of the sphere are determined by the boundary of the part. The preview image shows an elongated sphere in isometric view.

- Part Defined Shape: This will be either a cylinder, cube, or sphere as defined in the 3D model. Each part definition contains a simple shape and size. The parts display using their simple shape representation.

Preview

Displays an image which is a reasonable facsimile of the 3D shape with the current settings.

## Plan Tab (Structure Style Dialog Box)

Use this tab to define the appearance of the structure in 2D plan view.

### Structure

Specifies the method that will be used to define and draw the dimensions of the structure in 2D plan view.

**Use Outer Part Boundary**

The 2D plan view of the structure style is displayed as an outline of the 3D object specified on the Model tab. This option is enabled by default. A preview image is not available for this option.

**User Defined Part**

The structure style uses an AutoCAD block reference to display the structure. The Preview control displays the block listed in the Block Name drop-down list. The control lists all the blocks defined in the current drawing.

■ Block Name: Specifies the name of the drawing file to use as a structure symbol.

■ Select Drawing File Button: Click to select a drawing file to use as a structure symbol.

**Size Options**

When an AutoCAD block is selected above, the following size options are available:

■ Use Drawing Scale: When this option is selected, the specified value in the Units edit box (inches, or millimeters, when the drawing linear unit is set to Feet or Meters, respectively) is multiplied by the drawing scale to determine the size of the structure symbol displayed in the drawing. Use this setting when annotation symbols are used.

■ Use Size As Percentage Of Screen: When this option is selected, the specified value that is entered in the Units edit box is a percentage of the drawing screen size. The size of the structure symbol displayed in the drawing is always the percentage specified of the drawing screen size.

■ Use Size In Absolute Units: When this option is selected, the specified value that is entered in the Units edit box is an absolute value of the drawing linear units. The title text above this box displays Feet when the drawing units are set to feet, and Meters when the drawing units are set to meters.

■ Use Fixed Scale: When this option is selected the Fixed Scale group box is enabled. The structure symbol will be scaled by the Fixed Scale X, Y, and Z parameters.

■ Use Fixed Scale From Part Size: When this option is selected the Fixed Scale group box is enabled. The structure symbol will be scaled by the Fixed Scale X, Y, and Z parameters based on the object part size.

**Inch or Percent**

The type of units displayed above this box depends on what is selected in the Size options box.

■ Use Drawing Scale: When this option is selected, the title above this box displays as Inches when the drawing units are set to Feet, and Millimeters when the drawing units are set to Meters.

■ Use Fixed Scale: When this option is selected, the units box is disabled and the Fixed Scale X, Y, and Z boxes are enabled. The block size is based on the part size.

■ Use Size In Absolute Units: When this option is selected, the title above this box displays as Inches when the drawing units are set to Feet, and Millimeters when the drawing units are set to Meters.

■ Use Size As Percentage Of Screen: When this option is selected, the title above this box displays as Percent.

■ Use Fixed Scale From Part Size: When this option is selected, the units box is disabled and the Fixed Scale X, Y, and Z boxes are enabled. The block size is based on the part size. The default fixed scale factor for the block is 1 for X & Y.

**Enable Part Masking**

Specifies whether the pipe objects are masked by the plan structure object. This control is enabled by default.

**Preview**

When User Defined Part is selected, this area displays an image with the current setting.

# Profile Tab (Structure Style Dialog Box)

Use this tab to define the appearance of the 3D object used to represent the structure in profile view.

**Structure**

Specifies the method that will be used to define and draw the dimensions of the structure in profile view.

Display As Solid

The 3D object defined in the parts catalog is used when displaying the structure in 3D. For more information, Part Catalog and Parts Lists (page 1237). This option is enabled by default. A preview image is not available for this option.

Display As Boundary

Only the outer boundary of the structure, as defined in the parts catalog, is used when displaying the structure in a profile.

Display As Block

Specifies that the structure style uses an AutoCAD block reference to display the structure. The Preview control displays the block listed in the Block Name drop-down list. The control lists all the blocks defined in the current drawing.

- Block Name: Specifies the name of the drawing file to use as a structure symbol.

- Select Drawing File Button: Click to select a drawing file to use as a structure symbol.

Size Options

When an AutoCAD block is selected above, the following size options are available:

- Use Drawing Scale: When this option is selected, the specified value in the Units edit box (inches, or millimeters, when the drawing linear unit is set to Feet or Meters, respectively) is multiplied by the drawing scale to determine the size of the structure symbol displayed in the drawing. Use this setting when annotation symbols are used.

- Use Size As Percentage Of Screen: When this option is selected, the value specified in the Units edit box is a percentage of the drawing screen size. The size of the structure symbol displayed in the drawing is always the same percentage specified of the drawing screen size.

- Use Size In Absolute Units: When this option is selected, the specified value that is entered in the Units edit box is an absolute value of the drawing linear units. The title text above this box displays Feet when the drawing units are set to feet, and Meters when the drawing units are set to meters.

- Use Fixed Scale: When this option is selected the Fixed Scale group box is enabled. The structure symbol will be scaled by the Fixed Scale X, Y, and Z parameters.

- Use Fixed Scale From Part Size: When this option is selected the Fixed Scale group box is enabled. The structure symbol will be scaled by the Fixed Scale X, Y, and Z parameters based on the object part size.

Inch or Percent

The type of units displayed above this box depends on what is selected in the Size options box.

- Use Drawing Scale: When this option is selected, the title above this box displays as Inches when the drawing units are set to Feet, and Millimeters when the drawing units are set to Meters.

- Use Size Relative To Screen: When this option is selected, the title above this box displays as Percent.

- Use Size In Absolute Units: When this option is selected, the title above this box displays as Inches when the drawing units are set to Feet, and Millimeters when the drawing units are set to Meters.

- Use Fixed Scale: When this option is selected, the units box is disabled and the Fixed Scale X, Y, and Z boxes are enabled.

- Use Fixed Scale From Part Size: When this option is selected, the units box is disabled and the Fixed Scale X, Y, and Z boxes are enabled. The block size is based on the part size. The default fixed scale factor for the block is 1 for X & Y.

**Enable Part Masking**

Specifies whether the pipe objects are masked by the plan structure object. This control is enabled by default.

**Preview**

When User Defined Part is selected, this area displays an image with the current settings.

# Display Tab (Structure Style Dialog Box)

Use this tab to change the display and visibility of structure object components.

## View Direction

Some object styles can have unique display values and varying numbers of displayed components, depending on whether they are being displayed in Plan, Model, Profile, or Section views. You can create and set style characteristics for each supported view direction type using the View Direction list.

**Plan**

Specifies the display style settings when the structure is displayed in plan view.

**Model**

Specifies the display style settings when the structure is displayed in model view.

**Profile**

Specifies the display style settings when the structure is displayed in a profile view.

**Section**

Specifies the display style settings when the structure is displayed in a section view.

## Component Display

You can set the following display style characteristics for structure components: Visibility, Layer, Color, Linetype, LT Scale, and Lineweight.

**Structure**

The style for structures displayed in plan view.

**Structure Hatch**

The hatch pattern for structures.

**3D Solid**

The style for structures displayed in 3D model view.

**Structure Pipe Outlines**

The style for pipes that are displayed in profile or section views.

**Hydraulic Grade Line**

The style for displaying a hydraulic grade line for pipe networks that are displayed in profile views. For more information, see Viewing Hydraulic Data in Profile Views (page 1282).

**Energy Grade Line**

The style for displaying an energy grade line for pipe networks that are displayed in profile views. For more information, see Viewing Hydraulic Data in Profile Views (page 1282).

**Component Hatch Display**

In plan, profile, or section views, you can specify hatch display styles for the fill areas of the structure.

Structure Hatch

The hatch pattern for structures displayed in plan, profile, or section views.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Structure Style Dialog Box)

Use this tab to review all the information about the current structure style.

This information can be copied and pasted to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Part Size Creator Dialog Box

Use this dialog box to select or edit sizes and parameters for parts you wish to create.

The sizes and values available through this dialog box are defined from the pipe network part catalog. In most cases, there is only a single value available, or a drop-down list of choices, where you can select a value.

Property

Each part property is matched to the part type and may be selected, or added as an optional property, to a part size depending on how it was published in the part catalog. Part properties are characterized by value, units, and source.

Value

Specifies the current value defined for the property.

Units

Specifies the current units defined for this property, if applicable.

Source

Specifies how the property was published in the part catalog. For example, if the value is selected from a list, the source will be identified as Fixed List. Source types can be:

- Fixed List: This is a read-only, selectable list of fixed sizes or items from the catalog part family. Select the size from the drop-down list or check the all-sizes option to bring in all size combinations.

- Range: This is an editable value. The part catalog defines a min/max range and the user can enter any value within that range for part creation.

- Table: Values are arranged and selected in table rows. This is only used for pipes. Size selection is enabled for pipe diameter or width.

- Constant: A constant is a read-only data type. This is a value explicitly defined in the part family and cannot be changed.

- Calculation: This is a read-only value that is obtained from a calculation based on any other parameter or parameter(s).

- Optional Property: Lets you add one of the optional properties from a fixed list of optional properties for each defined part. For example an optional property called "Material" can be added to the part. Once the property is added to the part, the optional property then appears in the Part Size Creator dialog box where you can set its value. Optional properties can be used for labeling purposes. For more information, see Assigning Optional Properties to a Part Size (page 1242).

Add All SIzes

This option is available only to properties with a Fixed List source type. Selecting this options adds all of the part sizes available in the current property. Select this option if you wish to add all of the size options available for this property. For example, if you choose to Add All Sizes for a property called "Structure Height" (Fixed List values = 1.0, 2.0, and 3.0) and a property called "Headwall Top Width" (Fixed List values = 4.0 and 5.0), then the following set of part sizes are automatically added:

- Structure Height 1.0, Headwall Top Width 4.0

- Structure Height 2.0, Headwall Top Width 4.0

- Structure Height 3.0, Headwall Top Width 4.0

- Structure Height 1.0, Headwall Top Width 5.0

- Structure Height 2.0, Headwall Top Width 5.0

- Structure Height 3.0, Headwall Top Width 5.0

For more information, see Assigning Optional Properties to a Part Size (page 1242).

# Pipe Network Layers Dialog Box

Use this dialog box to define the layers used when creating pipe network components in plan, profile, and section views.

The default object layers are defined from the root drawing settings on the Object Layers tab.

Pipe Plan Layer

Specifies the layer used when creating pipe network pipes in plan view.

Structure Plan Layer

Specifies the layer used when creating pipe network structures in plan view.

Pipe Profile Layer

Specifies the layer used when creating pipe network pipes in a profile view.

Structure Profile Layer

Specifies the layer used when creating pipe network structures in a profile plan view.

Pipe Network Section Layer

Specifies the layer used when creating the pipe network in a section view.

# Select Parts List Dialog Box

Use this dialog box to select a parts list to associate with a pipe network.

The parts list contains the pipes and structures that are available for use in the current pipe network. You can create new parts lists or edit existing ones.

# Pipe Network Catalog Settings Dialog Box

Use this dialog box to configure AutoCAD Civil to access the imperial or the metric pipe network parts catalog, or to access specific pipe and structure catalogs within the imperial or metric catalog folders.

The part catalog settings defined here enable pipe network parts lists to access the content for pipe and structure shapes.

Part catalogs are set for each drawing. When in a drawing, and you switch between drawings, the catalogs (both pipe and structure) will automatically be set to the unit it was last in that particular drawing (either imperial or metric). You can alternate between drawings that use imperial or metric units without changing any settings.

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**NOTE** The default catalog unit is determined by the Drawing Units setting in the Drawing Settings dialog boxUnits and Zone tab. For new and previously existing drawings, use this setting on initialization to set the unit of measure. See the Specifying Drawing Settings (page 63) for more information.

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Catalog Folder
    Specifies the folder that contains the current AutoCAD Civil pipe network parts catalogs.

Pipe Catalog
    Specifies the current AutoCAD Civil pipe catalog.

Structure Catalog
    Specifies the current AutoCAD Civil structure catalog.

# Pipe Network Vistas

Use the vistas displayed in the Panorama window to view and edit objects that are part of the selected pipe network.

Using the Panorama window, you can quickly display all data associated with pipe network parts in one of two vistas. There is a tab for Pipe objects and a tab for Structure objects. You can select an object in the vista and view or edit the pipe network property data.

This window is similar to a spreadsheet. Each row represents a pipe or structure object in a pipe network, and each column represents a property for that object. You can re-size, move, and copy columns. You can also copy all or some of the data from a vista and paste it into a spreadsheet. For more information, see The Panorama Window (page 102).

**Pipe Properties**

Status
    Displays a check mark icon when all design criteria (rules) have been met for this pipe object. Displays a
    warning icon  to indicate if there are rule violations associated with this pipe object. The number indicates how many rules are broken. See the Rules tab (page 2043) on the Pipe Properties dialog box to see which rules are broken. Hovering over the icon displays a tooltip disrobing the condition.

Name
    Specifies the name of the pipe object.

Description
    Specifies the description for the pipe object.

Style
    Specifies the style for the pipe object.

Rule Set
    Specifies the rule set associated with the selected pipe object.

Render Material
    Specifies the render material for the pipe object.

Shape
    Specifies the shape description as defined from the part catalog definition for this part.

**Inner Diameter**

Specifies the value for the inner diameter of the pipe shape.

**Inner Width**

Specifies the value for the inner width of the pipe shape.

**Inner Height**

Specifies the value for the inner height of the pipe shape.

**Reference Alignment**

Specifies the object name of the alignment that is referenced by this pipe object.

**Start Offset**

Specifies the offset of the starting point for the pipe object. This offset is obtained from the alignment referenced by this pipe object. Flow direction is not considered.

**Start Station**

Specifies the station of the starting point for the pipe object. This station is obtained from the alignment referenced by this pipe object. Flow direction is not considered.

**End Station**

Specifies the station of the ending point for the pipe object. This station is obtained from the alignment referenced by this pipe object. Flow direction is not considered.

**End Offset**

Specifies the offset of the ending point for the pipe object. This offset is obtained from the alignment referenced by this pipe object. Flow direction is not considered.

**Reference Surface**

Specifies the object name of the surface that is referenced by this pipe object.

**Slope (Hold Start)**

Specifies the slope of the pipe from the pipe's starting point. Flow direction is not considered.

**Slope (Hold End)**

Specifies the slope of the pipe from the pipe's ending point. Flow direction is not considered.

**Slope**

Specifies the slope of the pipe (upstream or downstream) based on the flow direction.

**Start Structure**

Specifies the name of the structure object the pipe object is connected to at the pipe start point, not considering flow direction.

**Start Northing**

Specifies the northing of the pipe start point, not considering flow direction.

**Start Easting**

Specifies the easting of the pipe start point, not considering flow direction.

**Start Invert Elevation**

Specifies the invert elevation of the pipe start point, not considering flow direction.

**Start Centerline Elevation**

Specifies the centerline elevation of the pipe start point, not considering flow direction.

**Start Crown Elevation**

Specifies the crown elevation of the pipe start point, not considering flow direction.

**End Structure**

Specifies the name of the structure object the pipe object is connected to at the pipe's end point, not considering flow direction.

**End Northing**

Specifies the northing of the pipe end point, not considering flow direction.

**End Easting**

Specifies the easting of the pipe start point, not considering flow direction.

**End Invert Elevation**

Specifies the invert elevation of the pipe end point, not considering flow direction.

**End Centerline Elevation**

Specifies the centerline elevation of the pipe end point, not considering flow direction.

**End Crown Elevation**

Specifies the crown elevation of the pipe end point, not considering flow direction.

**2D Length - Center to Center**

Specifies the two-dimensional length of the pipe, using the center of the first attached structure as the pipe start point, and using the center of the end attached structure as the pipe end point. Some scenarios may require or prefer that pipe length be measured in this way, from the center of the first attached structure to the center of the next attached structure.

**3D Length - Center to Center**

Specifies the three-dimensional length of the pipe, using the center of the first attached structure as the pipe start point, and using the center of the end attached structure as the pipe end point. Some scenarios may require or prefer that pipe length be measured in this way, from the center of the first attached structure to the center of the next attached structure.

**2D Length - To Inside Edges**

Specifies the two-dimensional length of the pipe, using the inside edge of the first attached structure as the pipe start point, and using the inside edge of the end attached structure as the pipe end point. Some scenarios performing hydraulic quantity calculations may require or prefer that pipe length be measured in this way, from the inside edge of the first attached structure to the inside edge of the next attached structure.

**3D Length - To Inside Edges**

Specifies the three-dimensional length of the pipe, using the inside edge of the first attached structure as the pipe start point, and using the inside edge of the end attached structure as the pipe end point. Some scenarios performing hydraulic quantity calculations may require or prefer that pipe length be measured in this way, from the inside edge of the first attached structure to the inside edge of the next attached structure.

**Bearing**

Specifies the horizontal bearing for straight pipes, and chord bearing for curved pipes.

**Start Direction**

Specifies the bearing of the start point for curved pipes, not considering flow direction.

**End Direction**

Specifies the bearing of the end point for curved pipes, not considering flow direction.

**Radius**

Specifies the horizontal radius for curved pipes.

**Flow Direction**

Specifies the flow direction relative to how the pipe was drawn end to start or start to end. The flow is always assumed to be downstream unless Flow Method forces the flow otherwise.

**Flow Method**

Specifies whether the flow direction of a pipe is set by slope, start to end, or end to start. This property is used to force the flow direction of a pipe based on start and end points.

**Minimum Cover**

Specifies the minimum distance of cover for the pipe, measured from the top outside of the pipe to the referenced surface elevation.

**Maximum Cover**

Specifies the maximum distance of cover for the pipe, measured from the top outside of the pipe to the referenced surface elevation.

**Chord Length**

Specifies the horizontal chord length of a curved pipe, measured from pipe end to pipe end.

**Radius Point Northing**

Specifies the northing of the radius point of a curved pipe.

**Radius Point Easting**

Specifies the easting of the radius point of a curved pipe.

**HGL Up\Down**

Specifies the HGL Up and Down elevation values for the selected pipe object. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

**EGL Up\Down**

Specifies the EGL Up and Down elevation values for the selected pipe object. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

**Structure Properties**

**Status**

Displays a check mark icon when all design criteria (rules) have been met for this structure object. Displays a warning icon ![icon] to indicate if there are rule violations associated with this structure object. The number indicates how many rules are broken. See the Rules tab (page 2057) on the Structure Properties dialog box to see which rules are broken. Hovering over the icon displays a tooltip disrobing the condition.

**Name**

Specifies the name of the structure object.

**Description**

Specifies the description for the structure object.

**Style**

Specifies the style for the structure object.

**Rule Set**

Specifies the rule set associated with the selected structure object.

**Render Material**

Specifies the render material for the structure object.

**Type**

Specifies the type description as defined in the part catalog definition for this part. Valid choices are Junction, Inlet/Outlet, or Null.

Inner Diameter

Specifies the value for the inner diameter of the structure shape.

Inner Length

Specifies the value for the inner length of the structure shape.

Inner Width

Specifies the value for the inner width of the structure shape.

Inner Height

Specifies the value for the inner height of the structure shape.

Rotation Angle

Specifies the AutoCAD rotation angle of the structure object in the drawing.

Reference Alignment

Specifies the object name of the alignment that is referenced by this structure object.

Station

Specifies the station of the insertion point for the structure object. This station is obtained from the alignment referenced by this structure object.

Reference Surface

Specifies the object name of the surface that is referenced by this structure object.

Insertion Northing

Specifies the northing of the structure insertion point.

Insertion Easting

Specifies the easting of the structure insertion point.

Insertion Rim Elevation

Specifies the rim elevation of the structure insertion point. Typically, this applies to the rim for a junction structure.

Connected Pipes

Specifies the number of pipes connected to this structure.

Hydraulic Grade Line

Specifies the elevation of the hydraulic grade line for the selected structure object. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

Energy Grade Line

Specifies the elevation of the energy grade line for the selected structure object. For more information, see Adding Hydraulic Property Data to AutoCAD Civil Pipe Networks (page 1280).

**Related procedures:**

■ Editing Pipe Networks Using the Pipe Network Vistas (page 1204)

■ The Panorama Window (page 102)

■ Part Properties (page 1235)

# Swap Part Size Dialog Box

Use this dialog box to select the part size that will be used to replace the currently selected pipe network part.

Expand the desired part family, select a part size, then click OK.

You can swap a part with a part that is the same type (pipe to pipe or structure to structure), but is from a different part family and/or part size.

For example, a pipe can only be swapped with another pipe; a junction structure can only be swapped with another junction structure; an inlet/outlet structure can only be swapped with another inlet/outlet structure; and a null structure can only be swapped with a junction structure. You cannot swap a pipe with a structure, and vice versa. Only parts that are available to be swapped display in the Swap Part Size dialog box.

If you want to replace a part with a different part type, you must delete the part, and then add the new one.

Note that when swapping parts, connection elevations are maintained. For example, pipe invert elevations remain the same if a pipe is swapped with another pipe, or if a structure with one or more pipes attached to it is swapped with another structure.

**Related procedures:**

■ Swapping Pipe Network Parts (page 1214)

# Add Rule Dialog Box

Use this dialog box to add a rule to a part.

You can also view the parameter values and the name and location of the VBA macro project and macro name files associated with each rule. Click on the Parameter and or Value column names to sort the items in ascending or descending order.

Category
Specifies the category the currently selected rule, displayed in the Rule Name field, belongs to. Click to select a rule category.

Rule Name
Specifies the name of the rule currently selected. Only rules that belong to the currently selected Category are displayed in the drop down list. Click to select a rule.

Macro Project
Specifies the name of the VBA project containing the macro used for processing this rule.

Macro Name
Specifies the name of the macro used for processing this rule.

Rule Parameters
Displays the rule parameter values that are used by the VBA script to process the currently selected rule. On this dialog box, these values are read-only. You can click on the Parameter and or Value column names to sort the items in ascending or descending order. You can also right-click in a column or row to display a shortcut menu from which you can select options.

**Default Name Format**

Use these settings to specify the default name formats for pipe network components (pipe network, pipes, structures). Click ⬚ to open the Name Template dialog box, where you can specify the template. For more information, see Name Template Dialog Box (page 1806).

**Default Rules**

Use these settings to establish the default rule set assigned to pipes and structures that are added to the pipe network. Click ⬚ to open the Structure Default Rules dialog box, where you can specify the default rule set assigned to pipes and structures.

**Related procedures:**

■ Editing Part Rules (page 1258)

# Create Interference Check Dialog Box

Use this dialog box to create an interference check, defining a name for the interference check, an optional description, pipe networks, layer, and styles.

You can enable or change proximity check criteria for the interference check by clicking the 3D Proximity Check Criteria button.

Name

Specifies the name of the interference check. Each interference check must have a unique name.

Opens the Name Template (page 1806) dialog box, where you can modify the interference check naming template.

Description

Specifies an optional description for the interference check.

Network 1

Specifies the first pipe network that will be tested in this interference check.

Network 2

Specifies the second pipe network that will be tested in this interference check.

Layer

Opens the Pipe Network Layers (page 1985) dialog box, where you can assign the layer for displaying interference check results.

**Interference Style**

Style List

Displays the current style for interferences. Click the arrow to display the interference styles available in the drawing.

Style Selection

Specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from the drawing.

**Render Material**

Material List

Displays the currently used render material. Click the arrow to display the render material available in the drawing.

Render Material

Click to select a render material from the drawing.

3D Proximity Check Criteria

Click this button to display the Criteria (page 2071) dialog box. The Criteria dialog box lets you choose to run the interference check using a specified distance or a specified scale factor.

**Related procedures:**

■ Checking for Interferences (page 1226)

# Criteria Dialog Box

Use this dialog box to enable or disable proximity checking, and to specify a distance or scale factor for the proximity check.

For more information, see Proximity Checking Criteria (page 1228).

**Apply 3D Proximity Check**

This option is available for checking a single pipe network, or for comparing two pipe networks to determine if any parts from one network are too close to parts from another network.

When this option is selected, the interference check looks for and detects the proximity check conditions specified on this dialog box. When this option is not selected, the interference check does not check for the proximity conditions specified on this dialog box, and the proximity checking options are not available (grayed out). Whether proximity checking is enabled or disabled, the interference check still checks for parts that physically collide.

Use Distance

When this option is selected, you can enter a distance. When the interference check is run, any pipe network parts within the specified distance away from each other (on any side) are marked as an interference.

Use Scale Factor

When this option is selected, you can enter a value that is used as a scale factor. When you enter a scale factor value, AutoCAD Civil calculates the limits of the part (the pipe or the structure) multiplied by the scale factor value that is specified. For example, if the pipe diameter is 600mm, and you specify a scale factor value of 2, AutoCAD Civil identifies any interferences within a distance of 1200mm (or the pipe diameter, which is 600, multiplied by the scale factor value, which is two). When the interference check is run, any pipe network parts within that distance away from each other (on any side) are marked as an interference.

# Interference Check Properties Dialog Box

Use this dialog box to view or change the information associated with an interference check.

You can change properties such as style, layer on which the interference check object is placed, or the interference checking criteria.

# Information Tab (Interference Check Properties Dialog Box)

Use this tab to view or change the name, description, style information, and layer of an interference check.

Name

Specifies the name of the current interference check.

Description

Specifies an optional description for the current interference check.

**Interference Defaults**

Style

Specifies the current style option. Select other styles from the list or click to create a new style, copy or

edit the current style selection, or pick a style from drawing. Click  to open the Style Detail dialog box, where you can preview the style and creation information

Render Material

Specifies the default render material assigned to the interference check when the interference check is inserted into the drawing. Click  to select a render material from the drawing.

Layer

Specifies the drawing layer on which the interference check results are placed. Click  to open the Object Layer Dialog Box (page 1985) dialog box to select a different layer.

Show Tooltips

Controls whether tooltips are displayed for the interference check object(s) in the drawing.

## Criteria Tab (Interference Check Properties Dialog Box)

Use this tab to view or change the conditions set for an interference check.

### Apply 3D Proximity Check

This option is available for checking a single pipe network, or for comparing two pipe networks to determine if any parts from one network are too close to parts from another network.

When this option is selected, the interference check looks for and detects the proximity check conditions specified on this dialog box. When this option is not selected, the interference check does not check for the proximity conditions specified on this dialog box, and the proximity checking options are not available (grayed out).

For more information, see Proximity Checking Criteria (page 1228).

Use Distance

When this option is selected, you can enter a distance. When the interference check is run, any pipe network part that is less than the specified distance away from any other part in the specified network(s), in any direction, is marked as an interference.

Use Scale Factor

When this option is selected, you can enter a value that is used as a scale factor. When you enter a scale factor value, AutoCAD Civil calculates the limits of the part (the pipe or the structure) multiplied by the scale factor value that is specified. For example, if the pipe diameter is 600mm, and you specify a scale factor value of 2, AutoCAD Civil identifies any interferences within a distance of 1200mm (or the pipe diameter, which is 600, multiplied by the scale factor value, which is two). When the interference check is run, any pipe network parts within that distance away from each other (on any side) are marked as an interference.

## Statistics Tab (Interference Check Properties Dialog Box)

Use this tab to view statistical information on the current interference check.

### Comparison

Displays the names of the pipe networks used in the current interference check.

Network 1

Displays the name of the first pipe network used in the interference check.

Network 2

Displays the name of the second pipe network, if there is one, used in the interference check.

### Statistics

Displays general information for the current interference check, such as the number of interference instances found and if the interference check is still current (valid) or out of date (obsolete).

Number of Interferences

Displays the total number of pipe network parts that have met the specified interference criteria. Each pipe network part that meets any interference criteria is considered to be an interference.

Status

Displays the current state of the interference check. Current means that the interferences found still exist, and that no changes have been made to and of the pipe network parts included in the interference check. A status of out of date means that one or more of the parts included in the interference check have changed.

**NOTE** If any pipe network part included in the interference check has changed in any way, the interference check becomes out of date. Even changing the description of a pipe network part that is not in interference, but is part of the pipe network that was checked causes the interference check to become out of date.

# Interference Properties Dialog Box

Use this dialog box to view or change information on an interference object.

You can change properties such as name, description, and or style of an interference object.

## Information Tab (Interference Properties Dialog Box)

Use this tab to view or change the name, description, and or style information of an interference.

Name

Specifies the interference name for the current interference.

Description

Specifies an optional description of the current interference.

### Object Styles

Style

Specifies the current style option. Select other styles from the list or click to create a new style, copy or edit the current style selection, or pick a style from drawing. Click to open the Style Detail dialog box, where you can preview the style and creation information

Render Material

Specifies the default render material assigned to the interference when the interference is inserted into the drawing. Click to select a render material from the drawing.

Show Tooltips

Controls whether or not tooltips are displayed for the interference in the drawing.

## Statistics Tab (Interference Properties Dialog Box)

Use this tab to view statistical information, including specific location information, and part names of the parts involved in the interference.

### General

Displays general statistics for the two parts comprising the interference.

Network Part1 / Network1

Displays the object name for the first pipe or structure comprising the interference. This field also displays the name of the pipe network the pipe or structure belongs to.

Network Part2 / Network2

Displays the object name for the second pipe or structure comprising the interference. This field also displays the name of the pipe network the pipe or structure belongs to.

Interference Location(X, Y, Z)

Displays the X, Y, and Z location information for the interference.

Interference Check

Displays the name of the interference check this interference belongs to.

# Interference Style Dialog Box

Use this dialog box to control the display of interferences in a pipe network.

Create different styles for various types of interferences. For example, create one interference style that displays overlapping parts in red, and another that displays parts that are too close to one another in green.

In addition to color and layer settings, you can also choose

## Information Tab (Interference Style Dialog Box)

Use this tab to change the interference style name and description information, and to review details about the interference style, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

## View OptionsTab (Interference Style Dialog Box)

Use this tab to define the appearance of interferences in plan and model views.

### Symbol Options

Specifies the marker style that will be used to visually identify interferences in 2D plan view.

Marker Style

When this option is selected, a 2D marker is used to visually identify interferences in plan view.

### Solid Options

Specifies the method that will be used to visually identify interferences in model view.

Show True Interference Solid

When this option is selected, interferences appear as a solid of the true collision of the network parts. This includes solids that are created when using the proximity check options.

Show as Sphere

This is the default option. When this option is selected, interferences are identified in the drawing by a sphere that is inserted into the drawing at the location of the interference. This option can be useful for identifying interferences that are otherwise difficult to see, due to being obscured by other drawing entities. The size of the sphere can be defined by specifying the following size (diameter) options:

■ Diameter by True Solid Extents: When this option is selected, the sphere is inserted at the centroid of the interference solid, with a size equal to the extents of the true interference solid. This option may be useful to visually represent (symbolize) the severity of the collision between parts.

■ User Specified Diameter: The size of the sphere is determined by the value entered in the box. When Use Size in Absolute Units is selected, the value entered in the box is calculated in feet or meters, depending on drawing settings. When Use Drawing Units is selected, the value is calculated in inches or millimeters, depending on drawing settings.

## Display Tab (Interference Style Dialog Box)

Use this tab to change the display and visibility of an interference.

You can specify display styles for the following interference components:

**Plan 2D Symbol**: The style for the symbol that unidentified interferences displayed in 2D plan view. For example, if a rectangle is chosen as the symbol for identifying interferences in plan view, this component lets you set the display style for the rectangle.

**Model 3D Solid**: The style for interferences displayed in as solids in 3D model view.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Interference Style Dialog Box)

Use this tab to review all the information about the current interference style.

This information can be copied and pasted to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

## Match Elevation Dialog Box

Use this dialog box to set pipe elevations to consistent, matching values in cases where multiple pipes connect to a single structure.

This feature lets you quickly set pipe elevations in cases where multiple pipes connect to a single structure. This provides a method for quickly and accurately matching multiple pipe elevations to the crown, centerline, or invert elevation of a selected pipe. You can also add a drop value that lets you adjust that elevation value further, if desired.

For more information, see Matching Elevations on Connected Pipes (page 1219).

Drop Amount
> This value is added to the elevation of the pipe that is selected on this dialog box. By default, this value is zero. It gives you the ability to adjust the elevation you are matching to by a specified value.

## Pipe Table Creation Dialog Box

Use this dialog box to create a pipe table in a drawing.

Most of the controls in this dialog box are generic table creation controls. For a description of these controls, see Table Creation Dialog Box (page 2441).

Use the following controls to specify the points to be included in the pipe table:

By Network
> Specifies a pipe network to add to the pipe table.

Multiple Selection
> Selects the pipe networks to be added to the pipe table.

**Related procedures:**

- Adding Pipe Tables (page 1271)

# Structure Table Creation Dialog Box

Use this dialog box to create a structure table in a drawing.

Most of the controls in this dialog box are generic table creation controls. For a description of these controls, see Table Creation Dialog Box (page 2441).

Use the following controls to specify the structures to be included in the structure table:

**By Network**

Specifies a network. The structures within the selected network are added to the structure table.

**Multiple Selection**

Selects the multiple network structures to be added to the structure table. You can add structures from differing networks.

**Related procedures:**

■ Adding Structure Tables (page 1272)

# Table Cell Components Dialog Box

Use this dialog box to manage the text components within a given cell within a structure table.

This dialog box is accessed by double-clicking a cell in the Structure Table in the Table Style dialog box.

Component Name

Displays a list of existing table components defined for the table style. Select a component from this list to edit its properties.

Create Component

Creates a new table component. Select a component type from this list to create a new text component, or text-for-each component. The possible components are **Structure**, **Structure all pipes**, **Structure in flow pipes**, and **Structure out flow pipes**.

Copy Component

Copies the selected component.

Delete Component

Deletes the selected component.

Component Row Order

Displays the Component Row Order (page 2077)dialog box.

From the Create Component list, click to create a new text component.

**General**

Name

Specifies the name of the text component.
The default name, such as "Text.1," is the component name with a numeric increment. If the text component exists in a parent table style, then the name cannot be edited.

Visibility

Specifies whether the text component is visible in the label style.

# Component Row Order Dialog Box

Use this dialog box to determine the order of the text components within a structure table cell.

Use the Top and Bottom arrows to arrange the component row order.

## Component

Name

Displays the name of the text component.

# Rename Pipe Network Parts Dialog Box

Use this dialog box to rename connected parts (pipes and structures) in a pipe network.

You can select one or more connected parts in a pipe network and automatically rename them. You can enter a new name to be applied, or select a name template. Optionally, you can select a starting number to renumber the parts.

**NOTE** When using this command, if the first part you select is not connected to another pipe or structure, you can only rename that single part.

Rename Structures

Select this option to rename the selected structure(s).

## Structure Name Template

Name

Specifies the name that will be used for the selected structure(s).

Opens the Name Template (page 1806) dialog box, where you can modify the naming template that will be used for the selected structure(s).

Starting Number

This optional field lets you specify a starting number that will be used to number the selected structure(s).

Rename Pipes

Select this option to rename the selected pipe(s).

## Pipe Name Template

Name

Specifies the name that will be used for the selected pipe(s).

Opens the Name Template (page 1806) dialog box, where you can modify the naming template that will be used for the selected pipe(s).

Starting Number

This optional field lets you specify a starting number that will be used to number the selected pipe(s).

**Name Conflict Options**

You must select one of the following naming conflict options:

Skip Number

When this option is selected, if a name and number combination is already being used by a pipe network part in the drawing, then the number will be incremented until that name\number combination is available as a name for the selected part.

Rename Existing Parts

When this option is selected, if a name and number combination is already being used by a pipe network part in the drawing, then that part will be renamed so that the selected part can use that desired name instead. The original pipe network part will be renamed using the next available name and number combination available.

# Export to Storm Sewers Dialog Box

Use this dialog box to select the pipe network data in the drawing to export to an .stm file format.

This dialog box is used for exporting AutoCAD Civil pipe network data to an .stm file so that it can be used by the Hydraflow Storm Sewers Extension. For more information, see Exporting Pipe Network Data to an .stm File (page 1279).

**NOTE** You cannot open .stm files that were exported from AutoCAD Civil, or that were saved in the Storm Sewers Extension for Civil 3D 2010, in the Storm Sewers Extension for Civil 3D 2009, or earlier versions of the Storm Sewers Extension.

Specify Objects To Be Exported

Lists the AutoCAD Civil pipe networks that exist in the current drawing. Expand the collections to display all of the subcomponents.
Select or clear the check box to the left of each item select the items that you want to export to an .stm file format.

**NOTE** The check boxes have a tri-state display. If only some items are selected under a collection, the check box is shaded ☑. If all the items are selected, the check box is selected ☑. If all the items below the collection are cleared, the check box for the collection is cleared ☐.

Pick From Drawing

Specifies pipe networks to export by selecting from the drawing.

**NOTE** If you access this dialog box by right-clicking a collection in the Prospector tree and selecting Export to Storm Sewers, the Pick From Drawing button is not available.

OK

When you click OK, the Export Storm Sewers to File dialog box is displayed where you can select a location to save the .stm file.

**Related procedures:**

■ Exporting Pipe Network Data to an .stm File (page 1279)

# Part Matchup Settings Dialog Box

Use this dialog box to specify the part types that will be used when importing and exporting pipe networks between AutoCAD Civil and the Storm Sewers Extension.

## Import Tab (Part Matchup Settings Dialog Box)

Use this tab to view or change the AutoCAD Civil part types that will be matched up to Storm Sewers part types when importing pipe network data from a Storm Sewers Extension .stm file into AutoCAD Civil.

The list of available Storm Sewers part types is displayed in the first column. In the second column, click in a cell, and then click ⊡ to display the AutoCAD Civil Part Catalog dialog box. Select a part type and click OK.

**Related procedures:**

■ Setting Part Matching Defaults for Migration (page 1276)

■ Part Catalog Dialog Box (page 2037)

Importing Circular Structures

This section lets you specify the AutoCAD Civil part types that will be used when importing circular structures from a Storm Sewers Extension .stm file.

Importing Rectangular Structures

This section lets you specify the AutoCAD Civil part types that will be used when importing rectangular structures from a Storm Sewers Extension .stm file.

Miscellaneous Part Types

This section displays the AutoCAD Civil part types that will be used for importing Storm Sewer parts that do not have a part type specified. Storm Sewers parts that do not have a part type specified are by default matched up to null structures when imported into AutoCAD Civil.

Importing Pipes

This section lets you specify the AutoCAD Civil part types that will be used when importing pipes from a Storm Sewers Extension .stm file.

## Export Tab (Part Matchup Settings Dialog Box)

Use this tab to view or change the AutoCAD Civil part types that will be matched up to Storm Sewers part types when exporting pipe network data from AutoCAD Civil to an .stm file.

The list of availableAutoCAD Civil part types is displayed in the first column. In the second column, click in a cell to select the Storm Sewers part type, and click OK.

**Related procedures:**

■ Setting Part Matching Defaults for Migration (page 1276)

■ Part Catalog Dialog Box (page 2037)

Exporting Structures

This section lets you specify the Storm Sewers part types that will be used when exporting structures from AutoCAD Civil to an .stm file.

### Exporting Pipes

This section lets you specify the Storm Sewers part types that will be used when exporting pipes from AutoCAD Civil to an .stm file.

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# Plan Production Tools Dialog Boxes

The following topics describe the plan production tools dialog boxes.

## Edit Feature Settings - View Frame Group Dialog Box

Use this dialog box to view and change view frame group-specific settings.

This topic documents settings in all view frame group-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the  drawing icon.

■ View frame group feature settings are listed near the top of this dialog box, after the General property group, and are identified by the  view frame group icon.

■ View frame group command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### Default Styles

Use these settings to specify the default styles assigned to view frame group components.

Match Line Style

Specifies the default style for match lines. Click in the Value column, and click ⋯ to select a style in the Match Line Style dialog box.

View Frame Style

Specifies the default style for view frames. Click in the Value column, and click ⋯ to select a style in the View Frame Style dialog box.

**2081**

**Match Line Left Label Location**

Specifies the default location for labels that are displayed on the left side of a match line. Click in the value field to select a location from the list.

**Match Line Right Label Location**

Specifies the default location for labels that are displayed on the right side of a match line. Click in the value field to select a location from the list.

**View Frame Label Location**

Specifies the default location on the view frame for view frame labels. Click in the value field to select a location from the list.

**View Frame Label Style**

Specifies the default style for view frame labels. Click in the Value column, and click [...] to select a style in the View Frame Label Style dialog box.

**Match Line Label Style Left**

Specifies the default style for labels that are displayed on the left side of a match line. Click in the Value column, and click [...] to select a style in the Match Line Label Style Left dialog box.

**Match Line Label Style Right**

Specifies the default style for labels that are displayed on the right side of a match line. Click in the Value column, and click [...] to select a style in the Match Line Label Style Right dialog box.

**Profile View Style**

Specifies the default profile view style. Click in the Value column, and click [...] to select a style in the Profile View Style dialog box.

**Profile View Band Set**

Specifies the default style for profile view band sets. Click in the Value column, and click [...] to select a style in the Profile View Band Set dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use these settings to specify the default name formats for view frame group components (view frame groups, view frames, match lines, layouts, and sheet files). Click in the Value column, and click [...] to select a style in the Name Template dialog box (page 1806).

## Sheet Creation

**NOTE** This property group is displayed when accessing the settings from the CreateSheets command.

Use these settings to specify the default styles assigned to sheet creation.

**Sheet Creation Method**

Specifies the default method for creating new sheets:

- A specified number per new drawing

- All in a new drawing

- All in the current drawing

Number of Layouts per Drawing
Specifies the number of layouts per new drawing.

Align North Arrow
Specifies whether the selected North arrow block is aligned to North.

North Arrow Block Name
Specifies the name of the selected North arrow, as entered in the cell.

Sheet Set Use
Specifies whether a you create a New sheet set for any new sheets, or whether you Add To Existing sheet sets.

Existing Sheet Set
Specifies the sheet set file to which you add a new sheet, if you select Add To Existing for Sheet Set Use.

Add to Vault
Specifies whether newly created sheet set files are added to Vault.

Align Profile and Plan View
Specifies whether profile and plan views are aligned:

- At Start Station

- At Center

- At End Station

## View Frame Creation

**NOTE** This property group is displayed when accessing the settings from the CreateViewFrames command.

Use these settings to specify the default styles assigned to view framecreation.

Alignment Station Range
Specifies whether creation of view frames is Automatic for the entire alignment, or for a User Specified range.

Sheet Type
Specifies type of sheet to be created:

- Plan And Profile

- Plan Only

- Profile Only

View Frame Orientation
Specifies whether view frames orient Along Alignment or Rotate To North orientation.

Enable First View Frame Offset
Specifies whether the first view frame is offset.

First View Frame Offset Distance
Specifies the distance by which the first view frame is offset. Enter a distance in the Value column or click
and select a distance in the drawing area.

Insert Match Line
Specifies whether match lines can be inserted.

Match Line Snap Station

Specifies whether match lines snap down to a rounded station value.

Match Line Snap Station Value

Specifies a whole number, multiples of which are the station values.

Match Line Repositioning

Specifies whether match lines can be repositioned after placement.

Match Line Repositioning Value

Specifies the distance by which match lines can be repositioned after placement. Enter a distance in the Value column or click  and select a distance in the drawing area.

# Edit Feature Settings - View Frame Dialog Box

Use this dialog box to view and change standard drawing ambient settings for all view frame objects.

The standard drawing ambient settings are preceded with . For feature-specific settings for View Frame Style and the View Frame Name Template, see Edit Feature Settings - View Frame Group Dialog Box (page 2081).

For more information about this dialog box, see Working with the Standard Settings Dialog Box Controls (page 62).

# Edit Feature Settings - Match Line Dialog Box

Use this dialog box to view and change standard drawing ambient settings for all match line objects.

The standard drawing ambient settings are preceded with . For feature-specific settings for Match Line Style and the Match Line Name Template, see Edit Feature Settings - View Frame Group Dialog Box (page 2081).

For more information about this dialog box, see Working with the Standard Settings Dialog Box Controls (page 62).

# Create View Frames Wizard

Use this wizard to create a set of view frames that will be automatically placed along a selected alignment.

Using this wizard quickly leads you through the process of creating view frames along an alignment. The view frames created will be automatically associated with a view frame group. When you finish using this wizard, the view frame and view frame group objects are displayed in the drawing and in the Prospector tree. If you chose to insert match lines, the match line objects are also displayed in the drawing and in the Prospector tree.

For more information, see Creating View Frames (page 1703).

# Alignment Page (Create View Frames Wizard)

Use this page of the wizard to select the alignment and station range for creating view frames.

### Alignment

Select an alignment name in the list, or click  and select an alignment in the drawing area.

### Station Range

Specify a station range for the alignment that is currently selected in the Alignment field above.

Automatic

   Selects the entire alignment.

User specified

   When this option is selected, you may enter a value or click ⬚ to select a location in the drawing area.

If the Create View Frames button is unavailable (grayed out), it may be for one of the following reasons:

■   There are no templates available or selected.

■   There are no valid layouts in the selected template. For example, there are no appropriate viewports in the selected template.

**Related procedures:**

■   Creating View Frames (page 1703)

## Sheets Page (Create View Frames Wizard)

Use this page of the wizard to select the type of sheets to create, a template for the sheets, and to determine how view frame are aligned.

### Sheet Settings

Select the type of sheet to generate:

Plan And Profile

   This is the default choice. The sheets that will be created will contain both a plan view and a profile view.

Plan Only

   The sheets that will be created will contain plan views, but no profile views.

Profile Only

   The sheets that will be created will contain profile views, but no plan views.

Template For Sheet

   You must select the template to be used by the sheet. Browse to a template (.dwt) or enter a valid path to
   specify the template to be used. When you click ⬚
   to browse to a template, the Select Layout as Sheet Template (page 2100) dialog box is displayed.
   If the Create View Frames button is disabled on this page of the wizard, it may be because a template has not yet been selected in this field.

   **NOTE** If you select a template that does not contain appropriately defined viewports, a message is displayed on the Select Layout As Sheet Template dialog box indicating that no layouts contain the necessary viewports for the selected sheet type. For more information, see Configuring Viewports for Plan Production (page 1700).

### View Frame Placement

Specifies the orientation and placement of the view frame.

Along Alignment

   When this option is selected, the view frames will be aligned following along the alignment. Typically, this is the preferred method for making the most efficient use of paper when plotting.

Rotate To North

When this option is selected, the view frames will be rotated according to the north orientation in the drawing.

Set The First View Frame Before The Start Of The Alignment By

This option allows you to set a distance by which the first view frame is offset from the location on the alignment where it begins. Entering a distance value here adds a margin so that the start location of your view frame does not hide important drawing data that you want to display. If you leave this field blank, some drawing data near the start of the first view frame may be too close to the start of the view frame.

**Related procedures:**

■  Creating View Frames (page 1703)

## View Frame Group Page (Create View Frames Wizard)

Use this page of the wizard to specify criteria for creating the view frame group object.

### View Frame Group

Name

Specifies the name for the view frame group.

Opens the Name Template (page 1806) dialog box, where you can modify the view frame group naming template.

Description

Enter an optional text description identifying the view frame group.

### View Frame

View Frame Layer

Displays the layer on which the view frame will be created.

Opens the Object Layer dialog box, where you can select a different layer for the view frame.

Name

Specifies the name of the view frame.

**NOTE** To name the view frame, either click on its default name and enter a new name, or use the naming template.

Opens the Name Template (page 1806) dialog box, where you can modify the view frame naming template.

View Frame Style

Displays the style that is used to display certain components of the view frame, such as layer, line style, line weight, color, and so on.

Click to select the style of a view frame in the drawing.

View Frame Label Style

Displays the style that is used to display labels on view frames. For more information, see Understanding Labels (page 1466).

Click to select the label style of a view frame in the drawing.

Label Location

Specifies the location where the label will be placed on the view frame. For example, selecting Top Left places the label starting at the top left side of the view frame.

**Related procedures:**

■ Creating View Frames (page 1703)

## Match Lines Page (Create View Frames Wizard)

Use this page of the wizard to configure a variety of choices that determine how and if match lines will be placed on the view frames.

Insert Match Lines

Match lines are only displayed in Model space and in plan views. They are not displayed in profile views. If you have selected to create plan and profile or profile only sheets, this option is automatically selected and you cannot clear (edit) it.

When this option is selected, match lines will be inserted on the view frames in plan views. If you do not want match lines on the view frames, clear this check box.

### Positioning

Use these options to adjust where match lines are placed, and how they may be moved after creation, so that they do not obstruct data that you want to display.

Snap Station Value Down To The Nearest

Clear this option to use a rounding value (calculation) for the match line positioning based on derived stations. Select this option to use the rounding calculation value that is entered. The rounding calculation always rounds down. For example, if the calculated station for a match line is 48+37.69, then a rounding of 100 would place the match line on 48+00.

| When Snap Station Value Down To The Nearest is set to ... | the station will round to |
| --- | --- |
| 100 | 18+00 |
| 50 | 18+50 |
| 10 | 18+60 |
| 1 | 18+65 |
| 27+45.3 to 50 | 27+00 to the next lowest multiple of 50 |

**NOTE** This option will not accept values that cause the match lines to be placed in undesirable locations, such as before the previous match line or before the beginning of the alignment. If a rounding calculation would result in the match line being placed in an undesirable location, then the rounding calculation is ignored and the match lines are placed at calculated station.

Allow Additional Distance For Repositioning (Increases View Overlap)

This option gives you the ability to increase the distance that you can move match lines in plan view after they have been created. When you increase this margin, or additional distance, the overlap area of

the match lines increases. This option can also be used to force an overlap of frames in plan sheets only when, on a straight line alignment for example, adjacent edges of two view frames might be coincident, and therefore not provide enough room on either side of the match line for labeling and notes.

**Match Line**

Layer

Displays the layer that the match line will be created on.

Opens the Object Layer dialog box, where you can select a different layer for the match line.

Name

Specifies the name of the match line.

**NOTE** To name the match line, either click on its default name and enter a new name, or use the naming template.

Opens the Name Template (page 1806) dialog box, where you can modify the match line naming template.

Match Line Style

Displays the style that is used to display the match line and the match line mask area. For more information, see Match Line Style Dialog Box (page 2099) and/or Editing the Match Line Mask Area (page 1715).

Click to select the style of a match line in the drawing.

**Labels**

Left Label Style

Displays the style that is used to display the match line label that is on the left side of the match line.

Click to select the label style of a match line in the drawing.

Left Label Location

Specifies the location for the label that is placed on the left side of a match line. For example, selecting End places the left side match line label at the end of the match line.

Right Label Style

Displays the style that is used to display the match line label that is on the right side of the match line.

Click to select the label style of a match line in the drawing.

Right Label Location

Specifies the location for the label that is placed on the right side of a match line. For example, selecting End places the right side match line label at the end of the match line.

**Related procedures:**

■ Creating View Frames (page 1703)

## Profile Views Page (Create View Frames Wizard)

Use this page of the wizard to select the profile view style and band set that will be used for the profile views displayed in the viewports (sheets).

It is important to understand that the profile view style and band set for the profile view cannot be changed later when you are using the Create Sheets wizard to create sheets. For more information, see Understanding Profile View Options (page 1724).

Profile View Style

Specifies the style that is used to display the profile views in the viewports (sheets).

Band Set

Specifies the band set that is used to display the profile views in the viewports (sheets). You can copy an existing band set.

**Related procedures:**

■ Creating View Frames (page 1703)

## Create Sheets Wizard

Use this wizard to automatically create sheets from a view frame group.

Using this wizard quickly leads you through the process of creating sheets from a selected view frame group. The sheets created display the data that is inside each view frame within the view frame group. When you finish using this wizard, you may want to then use the Sheet Set Manager to create, organize, and manage sheet sets.

**See also:**

■ Creating Sheets (page 1719)

## View Frame Group and Layouts Page (Create Sheets Wizard)

Use this page of the wizard to select the view frame group and output settings for layout creation.

### View Frame Group

Specifies the view frame group used to create the sheets. Click  to select a view frame group in the drawing.

View Frame Range

Specifies which view frames to include in the sheets.

■ **All**: This is the default setting. Select this option to create sheets for every view frame in the view frame group.

■ **Selection**: When this is selected, the Choose View Frames button is available, and you can click it to select one or more view frames from the list in the Select View Frame dialog box.

### Layout Creation

These options determine how the layouts will be created.

Number Of Layouts Per New Drawing

This option may be an appropriate choice if you plan to have several people working on individual sheets. When this option is selected, and you enter 1 for the value, you get one new layout (sheet) created in each new drawing. The total number of sheets and drawings would depend on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template.

If you select this option and enter a value greater than one, three for example, three new layouts would be created in each new drawing. The total number of sheets and drawings would depend on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. You can enter an integer from 1 to 255.

All Layouts in One New Drawing

This option may be an appropriate choice if having each individual layout in a separate drawing is not necessary for you, and if you have less than 10 sheets. You may place all the layouts in one newly created drawing. The total number of sheets would depend on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. For optimal results, it is recommended that you create no more than 10 sheets per drawing. Therefore, this option may not be the best choice if you have more than 10 sheets.

All Layouts in The Current Drawing

If you are only generating less than 10 sheets, you may want to generate the layouts in the current drawing. The total number of sheets would depend on the length of the alignment selected and other criteria, such as the size of the viewports in the referenced template. For optimal results, it is recommended that you create no more than 10 sheets per drawing. Therefore, this option may not be the best choice if you have more than 10 sheets.

Layout Name

Specifies the name of the layout(s) that will be created.



Opens the Name Template (page 1806) dialog box, where you can modify the naming template for the layouts.

Choose The North Arrow Block To Align In Layouts

All blocks that exist in the currently selected template are displayed in the drop-down list. If you select a block from this list, that block will be aligned to the north on the sheet(s). Typically, this option is used for aligning a north arrow block in the layouts.

**Related procedures:**

■ Creating Sheets (page 1719)


# Sheet Set Page (Create Sheets Wizard)

Use this page of the wizard to specify creation criteria for the sheet set, such as names and locations for the sheet set, the sheet set file (*.dst*), and the sheet file.

Add Files To Vault

If you are currently logged in to a Vault, this field is available and automatically selected, and the following files, which are related to the sheet set creation, will be checked in to the Vault:

■ Sheet files created from this wizard session

■ Sheet set files created from this wizard session

This field is not available if you are not currently logged in to a Vault, or if Autodesk Vault is not installed.

Log in to Vault...

This button is not available (grayed out) if you are already logged in to a Vault, or if Vault is not installed on the machine. Clicking this button displays the Vault Log In dialog box which prompts you to log in to a Vault server and a Vault database. If you wish to use Vault to manage the files created during this wizard session, you must be logged in to Vault.

### Sheet Set

Use these options either to create a new sheet set or add to an existing sheet set. You can also specify the storage location of the sheet set file associated with the new or existing sheet set.

New Sheet Set

> Creates a new sheet set using the name of the view frame group displayed in the text box.

Add To Existing Sheet Set

> Creates sheets adds them to an existing sheet set. Click [...] to select a sheet set.

Sheet Set File (.DST) Storage Location

> Specifies the location where the sheet set file created or used during this session is located. Click [...] to select a location. If you selected Add to Existing Sheet Set, this field displays the location of the sheet set file for the sheet set that you selected in that field.

### Sheets

These options determine how the sheet files will be named and stored. If you have chosen to save all sheets in the current drawing (the All Layouts In The Current Drawing choice on the Create Sheets wizard View Frame Group and Layouts page), these fields are not available (read only, grayed out).

Sheet Files Storage Location

> Specifies the location where the sheet files created during this session will be located. Click [...] to select a location.
> If you are logged in to a project in Vault, by default this path displays the root folder for the project. If you are not logged in to a project in Vault, by default this path is set to the directory containing the working (current) drawing.

Sheet File Name

> Specifies the name of the sheet file(s) that will be created.

> 

> Opens the Name Template (page 1806) dialog box, where you can modify the naming template for the sheet files.

### Related procedures:

■ Creating Sheets (page 1719)

## Profile Views Page (Create Sheets Wizard)

Use this page of the wizard to view the profile view style and band set selected during view frame creation. While you cannot change the profile view style or the band set chosen, you can configure certain other profile view options.

If you want to change the profile view style or band set, click Cancel on the Create Sheets wizard, return to the Create View Frames wizard, change the profile view style and/or the band set there, recreate the view frames, and then use the Create Sheets wizard. Optionally, you can launch the Profile View Wizard from this page by choosing Choose Settings, and then clicking Profile View Wizard.... This lets you change a variety of profile view options. For more information, see Understanding Profile View Options (page 1724).

If you have chosen an output that does not include profiles, this page of the Create Sheets wizard is skipped (not displayed), and the Data References page of the Create Sheets wizard is displayed.

### Profile View Settings

Profile View Style To Be Used

This read-only field displays the profile view style that was chosen during frame view creation. This is the profile view style that will be added to each profile view.

Band Set To Be Used

This read-only field displays the band set that was chosen during frame view creation. This is the band set that will be used to generate these sheets.

### Other Profile View Options

Use these options to change certain settings on the profile views that will be used to create these sheets. For more information, see Understanding Profile View Options (page 1724). If you do not want to change any profile view settings, click Next.

Get Other Settings From An Existing Profile View

Select this option if you want to use the settings from another profile view in the current drawing. Select

a profile view in the list or click  to select a profile view in the drawing.

Choose Settings

Select this option if you want to use the Create Multiple Profile Views wizard to select options from multiple profile views. When this option is selected, the Profile View Wizard... button is available.

Profile View Wizard...

Click the Profile View Wizard... button to open the Create Multiple Profile View wizard. At this point in the Create Sheets wizard, you can use the Create Multiple Profile Views wizard to create multiple profile views if desired. If you use this option, multiple profile views will be created and placed in model space in the file containing the sheets. At the end of the Create Multiple Profile Views wizard session, you are returned to the Profile Views page of the Create Sheets wizard.

### Align Views

When creating plan and profile sheets, you can choose how to align the data that is displayed in the plan and profile views. If you are creating plan only or profile only sheets, these options are not available. These options are useful on projects that require the data displayed in sheets to be aligned according to certain project requirements (such as aligned left, center, or right).

Align Profile And Plan View At Start

The left side (start station) match line in the plan view aligns with the left side (start) of the profile view.

Align Profile And Plan View At Center

The center of the plan view aligns with the center point of the profile view.

Align Profile And Plan View At End

The right side (end station) match line in the plan view aligns with the right side (end) of the profile view.

**Related procedures:**

■ Creating Sheets (page 1719)

## Data References Page (Create Sheets Wizard)

Use this page of the wizard to select the data that you want referenced in your sheets.

In the collections that are displayed, items that are required for creating the sheets, such as the selected alignments and profiles, are already selected. You cannot edit these items because you cannot clear the check mark. All other items that are available are expanded in the tree. Select the items in the tree that you want to include in your sheets.

If you have chosen to have the sheets saved in the current file, (in other words, if you have chosen All Layouts In The Current Drawing on the View Frame Group And Layouts page of the Create Sheets wizard), then the Data References page of the Create Sheets wizard is skipped (not displayed).

 Pick From Drawing

Click to select the objects in the drawing.

Copy Pipe Network Labels To Destination Drawings

Selecting this option copies any pipe network labels to the destination drawing(s). If no pipe networks exist, this option is not available.

**Related procedures:**

■ Creating Sheets (page 1719)

# View Frame Properties Dialog Box

Use this dialog box to view the properties of a view frame, such as the view frame name, description, and style.

The View Frame Properties dialog box also lists the view frame group, which template was used, and whether a sheet has been created from the view frame.

**See also:**

■ Editing View Frames (page 1711)

## Information Tab (View Frame Properties Dialog Box)

Use this tab to view or change general information for the view frame using the following settings:

Name

Specifies the name of the current view frame.

Description

Specifies an optional description for the current view frame.

Object Style

Specifies the default view frame style used to display view frames. Select a style in the list or use the standard style selection tools. For more information about the standard style selection tools, see the Select Style (page 1805) dialog box.

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## View Frame Tab (View Frame Properties Dialog Box)

Use this tab to view the parameters of the current view frame, including the alignment name, view frame group, view frame number, orientation, and station values.

Alignment

This read-only field displays the alignment that was used to create the view frame group.

View Frame Group

This read-only field displays the name of the view frame group this view frame belongs to.

## View Frame Number

This read-only field indicates the order in which this view frame is displayed in the sequence of view frames in this view frame group. For example, the first view frame displayed in the view frame group has a view frame number of 1. If there are a total of eight view frames in the view frame group, the last view frame in the group would have a view frame number of 8. Note that the view frame number is not related to the number counter that may be included in the view frame naming template.

## View Frame Orientation

Displays the orientation of the view frame; either Rotate To North or Along Alignment, depending on which option was selected in the View Frame Placement section of the Sheets page (Create View Frames wizard).

## Start Station

Displays the station value at which the match line associated with this view frame begins.

## End Station

Displays the station value at which the match line associated with this view frame ends.

# Sheet Tab (View Frame Properties Dialog Box)

Use this tab to view the sheet settings of the current view frame.

## Sheet

This read-only field displays the path, filename, and layout name for the sheet file containing the layout (sheet), if it has been generated. Otherwise it displays No Sheet Created.

## Type

This read-only field displays the type of sheet (Plan And Profile, Plan Only, or Profile Only).

## Sheet Set

This read-only field displays the path where the sheet set file associated with this view frame is stored, if it has been generated. Otherwise it displays No Sheet Created.

## Template Source

This read-only field displays the path, filename, and layout name for the sheet file/layout that was used for the sheet creation.

# Match Line Properties Dialog Box

Use this dialog box to view or change the properties of a match line, such as the match line style, name, and description.

The Match Line Properties dialog box also lists the match line number and which alignment station it is associated with.

**See also:**

■   Editing Match Lines (page 1715)

# Information Tab (Match Line Properties Dialog Box)

Use this tab to view or change general information for the match line.

## Name

Specifies the name of the current match line.

Description

Specifies an optional description for the current match line.

Object Style

Specifies the default match line style used to display the match line and the match line mask area (page 1715). Select a style in the list or use the standard selection tools. For more information about the standard style selection tools, see the Select Style (page 1805) dialog box. For more information about the match line style components, see Match Line Style Dialog Box (page 2099).

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Match Line Tab (Match Line Properties Dialog Box)

Use this tab to view the parameters of the current match line, including the alignment name, match line number, the station value where the match line is located, and the name of the view frame group that this match line is associated with.

Alignment

Specifies the alignment that this match line is associated with.

Match Line Number

This read-only field indicates the order in which this match line is displayed in the sequence of match lines in this view frame group. For example, the first match line displayed in the view frame group has a match line number of 1. If there are a total of eight match lines in the view frame group, the last match line in the group would have a match line number of 8. Note that the match line number is not related to the number counter that may be included in the match line naming template.

Station

This read-only field displays the station value where this match line is located.

View Frame Group

This read-only field displays the name of the view frame group that this match line is associated with.

## View Frame Group Properties Dialog Box

Use this dialog box to view or change the properties of a view frame group.

These are the default settings that are used when new view frames are added to a view frame group.

## Information Tab (View Frame Group Properties Dialog Box)

Use this tab to view or change general information for the view frame group object.

Name

Specifies the name of the current view frame group.

Description

Specifies an optional description for the current view frame group.

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

**Related procedures:**

■ Creating View Frames (page 1703)

# View Frames Tab (View Frame Group Properties Dialog Box)

Use this tab to view and edit the parameters of the view frames associated with the current view frame group.

These are the default settings that are used when new view frames are added to a view frame group using the InsertViewFrame command.

**View Frame Group Defaults**

View Frame Style

Displays the style that is used to display the view frame.



Click to select the style of a view frame in the drawing.

View Frame Layer

Displays the layer that the view frame will be created on.



Click to open the Object Layer dialog box, where you can select a different layer for the view frame.

Name

Specifies the name of the view frame.

**NOTE** To name the view frame, either click on its default name and enter a new name, or use the naming template.



Click to open the Name Template (page 1806) dialog box, where you can modify the view frame naming template.

The view frames table contains the following columns:

Number

This read-only field displays the number assigned to each view frame in this view frame group.

Name

Specifies the name of the view frame. You can change the name by clicking in the field.

Description

Specifies an optional description of the view frame. You can change the description by clicking in the field.

Begin Station

This read-only field displays the station along the alignment where this view frame begins. If the alignment that was used to create this view frame is unavailable or has been deleted, Not Available is displayed.

End Station

This read-only field displays the station along the alignment where this view frame ends. If the alignment that was used to create this view frame is unavailable or has been deleted, Not Available is displayed.

End Station

This read-only field displays the station along the alignment where this view frame ends. If the alignment that was used to create this view frame is unavailable or has been deleted, Not Available is displayed.

Layer

Specifies the layer of the view frame. You can change the layer by clicking in the field.

**Style**

Specifies the style of the view frame. You can change the style by clicking in the field.

**Layout Path**

This read-only field displays the path and filename of the layouts that are used by the view frames in this view frame group. If it was not generated, then No Sheet Generated is displayed.

# Match Lines Tab (View Frame Group Properties Dialog Box)

Use this tab to view and edit the parameters of the match lines associated with the current view frame group.

These are the default settings that are used when new match lines are added to a view frame group using the InsertViewFrame command.

## View Frame Group Defaults

**Match Line Style**

Displays the style that is used to display the match line and the match line mask area. Editing the Match Line Mask Area (page 1715)



Click to select the style of a match line in the drawing.

**Match Line Layer**

Displays the layer that the match line will be created on.



Click to open the Object Layer dialog box, where you can select a different layer for the match line.

**Name**

Specifies the default name template used for new match lines.

> **NOTE** To name the match line, click on its default name and enter a new name or use the naming template.



Click to open the Name Template (page 1806) dialog box, where you can modify the match line naming template.

The match lines table contains the following columns:

**Number**

This read-only fields displays the number assigned to each match line in this view frame group.

**Name**

Specifies the name of the match line. You can change the name by clicking in the field.

**Description**

Specifies an optional description of the match line. You can change the description by clicking in the field.

**Station**

This read-only field displays the station along the alignment where this match line is placed. If the alignment that was used to create this view frame group is unavailable or has been deleted, Not Available is displayed.

**Layer**

Specifies the layer of the match line. You can change the layer by clicking in the field.

Style

Specifies the style of the match line. You can change the style by clicking in the field.

Sheet Left

This read-only field displays the filename and sheet name of the sheet that is on the left side of the match line.

Sheet Right

This read-only field displays the filename and the sheet name of the sheet that is on the right side of the match line.

# Profile Views Tab (View Frame Group Properties Dialog Box)

Use this tab to view the parameters of the profile view(s) that may be associated with the current view frame group.

### Profile View Style

Profile View Style To Be Used

This read-only field shows you the style that will be used to display the profile view(s) associated with this view frame group.

### Band Set

Band Set Style To Be Used

This read-only field shows you the style that will be used to display the band sets for profile view(s) associated with this view frame group.

# View Frame Style Dialog Box

Use this dialog box to make it easier to locate and work with view frames by changing their display.

The view frame style controls the appearance of view frames.

**See also:**

■ View Frame Styles and Display (page 1695)

# Information Tab (View Frame Style Dialog Box)

Use this tab to change the view frame style name and description information, and to review details about the style, such as when it was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Display Tab (View Frame Style Dialog Box)

Use this tab to change the display and visibility of view frames.

### View Direction

Some object styles can have unique display values and varying numbers of displayed components, depending on whether they are being displayed in Plan, Profile, Section, or Model views. You can create and set style characteristics for each supported view direction type using the View Direction list.

Plan

Specifies the display style settings when the view frame is displayed in plan view. For view frames, only Plan View Direction is supported.

### Component Display

View Frame Border

This is the closed polyline entity that represents the extents of a view frame. You can set the following style characteristics for this component: Visibility, Layer, Color, Linetype, LT Scale, and Lineweight.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (View Frame Style Dialog Box)

Use this tab to review all the information about the current view frame style.

You can copy and paste this information to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Match Line Style Dialog Box

Use this dialog box to make it easier to work with match lines by changing their display characteristics.

The match line style controls the appearance of the match line and the match line mask area.

**See also:**

- Match Lines Styles and Display (page 1697)
- Editing the Match Line Mask Area (page 1715)

## Information Tab (Match Line Style Dialog Box)

Use this tab to view or change the match line style name and description information, and to review details about the style, such as when it was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

## Display Tab (Match Line Style Dialog Box)

Use this tab to view or change the display and visibility of match lines and the match line mask area.

### View Direction

Plan

Some object styles have unique display values and varying numbers of displayed components, depending on whether Plan, Profile, Section, or Model is selected for View Direction. For match lines, only Plan View Direction is supported.

### Component Display

Lines

Specifies the style for match lines. Match lines are only displayed in plan views, not profile views. You can set the following style characteristics for match lines: Visibility, Layer, Color, Linetype, LT Scale, Lineweight, and Plot Style.

Match Line Mask

Specifies the style for the match line mask areas. Match line mask areas are only displayed in plan view and in paper space. The the match line mask area is the area outside of match lines, between the outside of the match line, and the inside of the view frame boundary.

**Component Hatch Display**

Match Line Mask

Specifies the hatch pattern for the match line mask areas, which are only displayed in plan views, not profile views. You can define a pattern, angle and scale for hatching in this closed polyline area. For more information, see The Match Line Object (page 1696) and Editing the Match Line Mask Area (page 1715).

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Match Line Style Dialog Box)

Use this tab to review all the information about the current match line style.

You can copy and paste this information to the clipboard. For more information, see Summary Tab (Style Dialog Box) (page 1803).

## Select Layout as Sheet Template Dialog Box

Use this dialog box to select the template to use when you create a view frame group using the Create View Frames wizard.

The choices you make in this dialog box define the folder path, drawing template (DWT) file, and a layout name that will be used for creating the new sheets. You must choose a template that has viewports defined appropriately. For example, if you want to choose a template that contains viewports for both plan and profile views, you must select a template that contains viewports with Viewport Types that are defined as "plan" and "profile".

If you try to select a template that does not have appropriately defined viewports for the sheet type you selected on the Sheets page of the Create View Frames wizard (Plan Only, Profile Only, or Plan and Profile), AutoCAD Civil detects this and displays the following message: No layouts contain the necessary viewports for this sheet type.

For more information, see Configuring Viewports for Plan Production (page 1700).

Drawing Template File Name

Specifies the drawing template file path and name that will be used for creating the new sheets. Click ⋯ to select a valid template for the sheet(s) that will be created later when using the Create Sheets wizard.

Select A Layout To Create New Sheets

Lists and specifies the name of the layout in the drawing template file to be used for the new sheet(s) that will be created later when using the Create Sheets wizard.

**Related procedures:**

■ AutoCAD Help topics about Creating and Managing a Sheet Set

## Edit Match Line Group Labels Dialog Box

Use this dialog box to change label styles and locations for all match lines in a view frame group.

Left Match Line Label Style

Specifies the style for the labels that are displayed on the left side of a match line. You can choose a label style from the list.

Right Match Line Label Style

Specifies the style for the labels that are displayed on the right side of a match line. You can choose a label style from the list.

 Style Selection

Specifies the label style options. You can create a new label style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Left Label Location

Specifies the location for labels that are displayed on the left side of a match line. Click in the value field to select a location from the list.

■ Start: Places the left match line label at the start of the match line, near the top of the view frame.

■ End: Places the left match line label at the end of the match line, near the bottom of the view frame.

■ Middle: Places the left match line label at the middle of the match line.

■ Alignment Intersection: Places the left match line label where the alignment intersects the match line.

Right Label Location

Specifies the location for labels that are displayed on the right side of a match line. Click in the value field to select a location from the list.

■ Start: Places the right match line label at the start of the match line, near the top of the view frame.

■ End: Places the right match line label at the end of the match line, near the bottom of the view frame.

■ Middle: Places the right match line label at the middle of the match line.

■ Alignment Intersection: Places the right match line label where the alignment intersects the match line.

**Related procedures:**

■ Labeling Match Lines (page 1718)

# Edit Match Line Labels Dialog Box

Use this dialog box to change label styles and locations for a match line.

Match Line Label Style

Specifies the style for the labels that are displayed on a match line. You can choose a label style from the list.

 Style Selection

Specifies the label style options. You can create a new label style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Label Location

Specifies the location for labels that are displayed on a match line. Click in the value field to select a location from the list.

- Start: Places the match line label at the start of the match line, near the top of the view frame.

- End: Places the match line label at the end of the match line, near the bottom of the view frame.

- Middle: Places the match line label at the middle of the match line.

- Alignment Intersection: Places the match line label where the alignment intersects the match line.

**Related procedures:**

- Labeling Match Lines (page 1718)

# Edit View Frame Labels Dialog Box

Use this dialog box to change label styles and locations for a view frame.

View Frame Label Style

Specifies the style for the labels that are displayed on a view frame. You can choose a label style from the list.

 Style Selection

Specifies the label style options. You can create a new label style, copy or edit the current style selection, or pick a style from the drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

Label Location

Specifies the location for labels that are displayed on a view frame. Click in the value field to select a location from the list. For example, choosing Top Left places the view frame labels at the top left side of the view frame.

**Related procedures:**

- Labeling View Frames (page 1713)

# Points Dialog Boxes

# 63

Use the following links to access information about the Points dialog boxes.

## Edit Feature Settings - Point Dialog Box

Use this dialog box to view and change point-related settings.

This topic documents settings in all point-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the ▭ drawing icon.

- Point feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ✛ point icon.

- Point command settings are identified by the ☑ command icon.

You can also change the settings at the command level in the Create Points dialog box.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### ✛ Default Styles

Use these settings to establish the default styles assigned to points and points labels.

---

**IMPORTANT** By default, in the Create Points dialog box - Points Creation section the Disable Description Keys value is set to false and the controls in the Default Styles section are not available. To enable the Default Styles controls, set the Disable Description Keys value to true.

---

Point Style

Specifies the default point style. Click in the Value column, and click ⋯ to select a style in the Point Style dialog box.

Point Label Style

Specifies the default point label style. Click in the Value column, and click ![...] to select a style in the Point Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## ⊕ Default Name Format

Use these settings to specify the default name formats for new points and point groups. Click in the Value column, and click ![...] to make changes in the Name Template dialog box (page 1806).

## ⊕ Update Points

Use this setting to specify whether you can change local copies of project points without checking them out.

---

**WARNING** If you edit local copies of project points in a drawing without checking them out, you cannot check the points in to the project. Your changes exist only in the drawing in which you made them.

---

Allow Checked-In Points To Be Modified

Specifies whether you can change checked in points when you are working offline.

For more information, see Editing the Update Points Settings (page 412).

## ☑ Point Identity

Use these settings to assign defaults for point number identity.

Next Point Number

Specifies the next available point number.

Use Sequential Numbering

Specifies whether points are automatically numbered in sequence as they are created.

- False: Prompts you to assign point numbers as you create points.

- True: Automatically numbers points in sequence, starting with the number specified in Next Point Number

Point Number Offset

Specifies the offset added to imported point numbers when If Point Numbers Are Supplied is set to Add An Offset. Enter an offset.

Sequence Point Numbers From

Specifies the starting point number for imported points when If Point Numbers Need To Be Assigned is set to Sequence From. Enter a starting point number.

If Point Numbers Are Supplied

Specifies how to handle point numbers that are included in the point data being imported:

- Use: Uses the point numbers in the point data file to create points during import.

- Ignore: Does not use the point numbers in the point data file to create points during import. Points are numbered according to the If Point Numbers Need To Be Assigned setting.

- Add an Offset: Adds a fixed value, the value specified in Point Number Offset, to each point number in the point data file. For example, if you specify 200 for Point Number Offset, then points with numbers 1, 2, and 3 in the point data file are numbered 201, 202, and 203 when imported.

Force Names

Specifies if null point names are allowed.:

■ False: Point names are not automatically assigned to a point when it is created or imported.

■ True: Point names are always assigned to a point when it is created or imported.

If Point Numbers Already Exist

Specifies how to handle point numbers that already exist when the same point number is present in the data being imported:

■ Renumber: Assigns new point numbers to any imported points whose point numbers already exist. Points are numbered according to the If Point Numbers Need To Be Assigned setting.

■ Merge: Merges the point data of the existing point with the point data of the imported point. Point properties, such as elevation or raw description, that are specified in the point data file overwrite the point properties existing in the drawing. Point properties that are not specified in the point data file are preserved in the drawing.

■ Overwrite: Overwrites all point properties using the values in the point data file. No data is preserved; point properties that are not specified in the point data file are blank in the drawing.

■ Notify: Notifies you that a point number collision has occurred. Opens the Duplicate Point Number (page 2134) dialog box where you can specify how to resolve the collision.

NOTE For more information about how the Merge and Overwrite options work, see Editing the Point Identity Settings (page 413).

If Point Names Already Exist

Specifies how to handle point names that already exist when the same point name is present in the data being imported:

■ Use Name Template: Uses the name specified in the Point Name Template.

■ Counter: Increments a counter to the current point name.

■ Notify: Notifies you that a point name collision has occurred. Opens the Duplicate Point Name (page 2135) dialog box where you can specify how to resolve the collision.

If Point Numbers Need To Be Assigned

Specifies how new point numbers are assigned, when required, during import:

■ Use Next Point Number: Points are created using the next available point number.

■ Sequence From: Points are created beginning with the point number specified in Sequence Point Numbers From. Additional points are created with point numbers specified by incrementing the starting point number. For example, if you specify 300 for Sequence Point Numbers From, the first point is created with point number 300, the second point is created with point number 301, and the next point is created with point number 302.

## Table Creation

NOTE This property group is displayed when accessing the settings from the AddPointTable command.

Use these settings to establish the defaults when you add a point table.

Table Style

Specifies the style for a table. Click in the Value column, and click  to select a style in the  Table Style dialog box.

Split Table

Specifies whether a table is split into two or more sections after a specified maximum number of rows has been met.

Maximum Number of Rows

Specifies the maximum number of rows to include per section. If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of sections to include in each stack.

Table Spacing

Specifies the spacing between tables.

Tile Direction

Specifies the direction in which the table tiles (across or down).

## Default Layer

**NOTE** This property group is displayed when accessing the settings from the ConvertPointsToSdskPoints and CreatePoints commands.

Use this setting to establish default layer information when you create points.

Layer

Specifies a default layer. Click in the Value column, and click [...] to select or create a layer in the Layer Selection dialog box (page 1986).

## Points Creation

**NOTE** This property group is displayed when accessing the settings from the CreatePoints command.

Use these settings to establish default information for creating points.

Local Coordinates

Specifies the order and format used to display local coordinates on the command line during point creation:

- Easting - Northing

- Northing - Easting

- X - Y

- Y - X

Grid Coordinates

Specifies the order in which grid coordinates are displayed on the command line during point creation:

- Grid Easting - Grid Northing

- Grid Northing - Grid Easting

Geographic Coordinates

Specifies the order in which geographic coordinates are displayed on the command line during point creation:

- Latitude - Longitude

- Longitude - Latitude

**Prompt For Elevations**

Specifies how elevations are assigned point creation:

- Automatic: Creates points using the elevation value specified in Default Elevation.

- Manual: Prompts you to enter an elevation at the command line when a point is created.

- None: Creates points without elevations.

> **NOTE** If you are prompted for an elevation while creating a point and you want to create a point without an elevation, enter a period (.) at the prompt

**Prompt For Point Names**

Specifies how point names are entered during point creation:

- Automatic: Point names will automatically be assigned based on the point name template.

- Manual: Prompts for a point name at the command line when a point is created.

- None: Creates points without point names.

**Prompt For Descriptions**

Specifies how descriptions are assigned during point creation:

- Automatic: Creates points using the description specified in Default Description.

- Automatic-Object: Creates point description from the object.

  > **NOTE** Use the Automatic - Object option with commands from the Creating Points Based on Horizontal Alignments (page 459) list. When you specify this option the Alignment name and Station display in the Raw Description column in the Point Editor window.

- Manual: Prompts for a description at the command line when a point is created.

- None: Creates points without descriptions.

> **NOTE** If you are prompted for a description while creating a point and you want to create a point without a description, enter a period (.) at the prompt.

**Default Elevation**

Specifies the elevation automatically assigned by the point creation commands when the Prompt For Elevations setting is set to Automatic. Enter an elevation in the Value column or click ![icon] and select an elevation in the drawing area.

**Default Description**

Specifies the description that is automatically assigned by the point creation commands when Prompt For Descriptions is set to Automatic.

**Match On Description Parameters ($1, $2, Etc.)**

Specifies whether parameters are used in description key matching to rotate and scale the point symbol. For more information, see Controlling Whether Raw Description Values Are Interpreted as Parameters During Point Creation (page 539).

**Disable Description Keys**

Specifies whether description key matching occurs during point creation. For more information, see Activating Description Key Matching (page 539).

> **NOTE** If you change this setting, then the Disable Description Keys check box on the Create Points dialog box is also changed to the same state.

Echo Coordinates To Command Line

Specifies whether coordinates are displayed on the command line when creating points.

**Related procedures:**

■ Point Settings (page 410)

# Point Group Properties Dialog Box

Use this dialog box to change the properties of a point group.

## Information Tab (Point Group Properties Dialog Box)

Use this tab to view or change general information for the point group.

Name

Specifies the point group name. Change the number and the increment display in the Point Feature Settings. For more information, see Working with the Standard Settings Dialog Box Controls (page 62).

Description

Specifies a description for the point group.

Default Styles

Point Style

Specifies the default point style used to display a point in the point group, if no other style takes precedence. For more information, see Point Group Default Styles (page 523).

**NOTE** Changing the default point style updates the point style on the Overrides tab.

Point Label Style

Specifies the default point label style used to display a point in the point group, if no other style takes precedence. For more information, see Point Group Default Styles (page 523).

**NOTE** Changing the default point label style updates the point label style on the Overrides tab.

Object Locked

Specifies whether changes can be made to the point group in the drawing. Select the check box to lock the point group and prevent changes. For more information, see Locking and Unlocking Point Groups (page 528).

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

For more information about how point group styles are used to display a point, see Controlling the Appearance of Points in a Drawing (page 425).

**Related procedures:**

■ Creating Point Groups (page 529)

## Point Groups Tab (Point Group Properties Dialog Box)

Use this tab to include points in a point group by specifying one or more other point groups.

Point Group

Lists all point groups in the drawing. Select the check box next to the name of any point group you want to include in the current point group.

Description

Lists point group descriptions.

Information specified on this tab also appears on the Query Builder tab.

---

**NOTE** If an included point group is out-of-date, any point group that includes it is also out-of-date. For more information, see Out-of-Date Point Groups (page 524).

---

**Related procedures:**

■ Creating Point Groups (page 529)

## Raw Desc Matching Tab (Point Group Properties Dialog Box)

Use this tab to include points in a point group using raw description matching based on available description key codes.

This tab lists the description key codes for all the description keys in the drawing. When you select the check box next to a description key code, any point with a raw description that matches that code is included in the point group.

Code

Lists description key codes for all the description keys in the drawing. Select the check box next to the name of the description key codes you want to match.

Format

Specifies the format for the description key whose code is listed.

For more information about description keys, see Understanding Description Keys (page 537).

Information specified on this tab also appears on the Query Builder tab.

---

**NOTE** You can include or exclude a point from a point group based on its raw or full description. For more information, see Include Tab (Point Group Properties Dialog Box) (page 2109) and Exclude Tab (Point Group Properties Dialog Box) (page 2110).

---

**Related procedures:**

■ Creating Point Groups (page 529)

## Include Tab (Point Group Properties Dialog Box)

Use this tab to include points in a point group by point number, description, name, elevation, or by selection.

With Numbers Matching

Includes a point in the point group based on its point number or by selection:

■ Enter point numbers or ranges of point numbers separated by commas. Specify a range of point numbers by separating the beginning and ending numbers with a hyphen (for example, 100-105).

■ Click Selection Set In Drawing and select points from the drawing using standard AutoCAD selection techniques.

■ Click Project Points In Window and define a window by specifying two points in the drawing. The project northing and easting coordinates of the window corners are used to determine which project points are contained in the window. All points in the project to which the drawing is attached, and which fall within the window, are included in the point group.

**With Elevations Matching**

Includes a point in the point group based on its elevation. Enter any combination of the following, separated by commas:

■ An individual elevation

■ A greater than sign (>) followed by an elevation. Includes all points with an elevation above the specified value

■ A less than sign (<) followed by an elevation. Includes all points with an elevation below the specified value

■ An elevation range, which is specified by separating the beginning and ending numbers with a hyphen (for example, 1-100). Includes all points with an elevation that falls within the range

For example, <-100,1-100,110.01,>200 would include all points whose elevations meet one of the following criteria: less than -100, equal to or between 1 and 100, equal to 110.01, or greater than 200.

**With Names Matching**

Includes a point in the point group based on its name. Enter one or more point names separated by commas.

**With Raw Descriptions Matching**

Includes a point in the point group based on its raw description. Enter one or more raw descriptions separated by commas, for example IP**\***,GB**\***,TREE,STA.
The asterisk (**\***) is a wild card that matches any string. For example, the code IP**\*** will match all raw descriptions that begin with IP, so the raw description IPS matches the description IP**\***.

**With Full Descriptions Matching**

Includes a point in the point group based on its full description. Enter one or more full descriptions separated by commas, for example IP**\***,GB**\***,TREE,STA.
The asterisk (**\***) is a wild card that matches any string. For example, the code IP**\*** will match all full descriptions that begin with IP, so the full description IPS matches the description IP**\***.

**Include All Points**

Includes all points in the drawing in the point group. When the check box is selected, all other options on the tab are disabled.

Information specified on this tab appears on the Query Builder tab.

**Related procedures:**

■ Creating Point Groups (page 529)


# Exclude Tab (Point Group Properties Dialog Box)

Use this tab to exclude points from a point group by point number, description, name, elevation, or by selection.

#### With Numbers Matching

Excludes a point from the point group based on its point number or by selection. Select the check box and do one of the following:

■ Enter point numbers or ranges of point numbers separated by commas. Specify a range of point numbers by separating the beginning and ending numbers with a hyphen (for example, 100-105).

■ Click Selection Set In Drawing and select points from the drawing using standard AutoCAD selection techniques.

■ Click Project Points In Window and define a window by specifying two points in the drawing. The northing and easting coordinates of the window corners are used to determine which project points are contained in the window. All points in the project to which the drawing is attached that are within the window are excluded from the point group.

#### With Elevations Matching

Excludes a point from the point group based on its elevation. Enter any combination of the following, separated by commas:

■ An individual elevation

■ A greater than sign (>) followed by an elevation. Excludes all points with an elevation above the specified value

■ A less than sign (<) followed by an elevation. Excludes all points with an elevation below the specified value

■ An elevation range, which is specified by separating the beginning and ending numbers with a hyphen (for example, 1-100). Excludes all points with an elevation that falls within the range

For example, <-100,1-100,110.01,>200 would exclude all points whose elevations meet one of the following criteria: less than -100, equal to or between 1 and 100, equal to 110.01, or greater than 200.

#### With Names Matching

Excludes a point from the point group based on its name. Enter one or more point names separated by commas.

#### With Raw Descriptions Matching

Excludes a point from the point group based on its raw description. Enter one or more raw descriptions separated by commas, for example IP**,**GB**,**TREE,STA.
The asterisk (**) is a wild card that matches any string. For example, the code IP** will match all raw descriptions that begin with IP, so a raw description IPS matches the description IP**.

#### With Full Descriptions Matching

Excludes a point from the point group based on its full description. Enter one or more full descriptions separated by commas, for example IP**,**GB**,**TREE,STA.
The asterisk (**) is a wild card that matches any string. For example, the code IP** will match all full descriptions that begin with IP, so a full description IPS matches the description IP**.

Information specified on this tab appears on the Query Builder tab.

**Related procedures:**

■ Creating Point Groups (page 529)

## Query Builder Tab (Point Group Properties Dialog Box)

Use this tab to define point groups by using a query, which is a set of complex parameters or conditions.

A query consists of one or more expressions. Each row on the Query Builder tab contains an expression. You combine expressions using the set operators AND, OR, and NOT. You can also use parentheses to group expressions. For more information, see Understanding Point Group Queries (page 533).

When you first display the Query Builder tab, the starting query reflects the current contents of the following tabs: Raw Desc Matching, Include, Exclude, and Point Group.

This tab is read only until you select the Modify Query check box. To modify a cell in the query builder grid, click once in the cell to activate the row, then click again to activate the cell. To create a new row, right-click in the Query Builder and click Insert Row. To delete a row, click the row and press Delete.

---

**NOTE** After you select the Modify Query check box, the Raw Desc Matching, Point Groups, Include, and Exclude tabs become inactive. If you want to use these tabs, you must clear the Modify Query check box. For more information, see Creating Point Groups (page 529).

---

Use Case-Sensitive Matching
Specifies whether raw description matching and full description matching are case-sensitive, not only on this tab, but also on the Raw Desc Matching, Include, and Exclude tabs.

Modify Query
Controls whether the query builder tab is active.
Select this check box to edit the current query, which is derived from the contents of the Point Groups, Raw Desc Matching, Include, and Exclude tabs. These tabs become inactive, and their contents are lost, if you click Apply on the active Query Builder tab.
Clear this check box to activate the Point Groups, Raw Desc Matching, and Exclude tabs. If you click any of the activated tabs, the current query in the Query Builder is lost.

Set Operator
Specifies the set operator for an expression. Activate the cell and select AND, OR, or NOT. For more information, see Understanding Point Group Queries (page 533).

(
Turns opening parenthesis on or off. Activate the cell and click in it. For more information about using parentheses to combine expressions, see Understanding Point Group Queries (page 533).

Property
Specifies the property for the expression:

■ Name

■ Raw Description

■ Full Description

■ Point Number

■ Point Elevation

■ Point Group

Operator
Defines the relationship between Property and Value. The operators displayed in the list depend on the property you specified for the expression. Valid values include:

■ = (equal)

■ != (not equal)

■ > (greater than)

■ < (less than)

- \>= (greater than or equal)

- <= (less than or equal)

Value

Specifies the value for the property. Activate the cell and enter the value. Valid values depend on the property, as follows:

- Name: Enter a point name.

- Raw Description: Enter a raw description. You can use wild cards.

- Full Description: Enter a full description. You can use wild cards.

- Point Number: Enter an individual point number or a point range. A range is two point numbers separated by a hyphen, such as 10-21.

- Point Elevation: Enter an individual elevation or a range. A range is two elevations separated by a hyphen, such as 1001-1050.

- Point Group: Enter a point group name.

)

Turns closing parenthesis on or off. Activate the cell and click. For more information about using parentheses to combine expressions, see Understanding Point Group Queries (page 533).

Reset

Resets the current query to the starting query. Any changes you made since you entered the query builder are lost.

Apply

Applies the current query to the point group and updates the Point List and Summary tabs.

**Related procedures:**

- Creating Point Groups (page 529)

## Overrides Tab (Point Group Properties Dialog Box)

Use this tab to override some properties for the points in a point group.

Override values specified in this dialog box are displayed in the drawing, the point group item view, and on the Point List tab of the Point Group Properties dialog box. However, the point itself is not changed. The point group display order can affect the override values displayed in labels for raw description and elevation. It can also affect both the style and label style that is used to display a point in a drawing. For more information, see Using Point Groups to Override Point Properties (page 523).

Raw Description

Overrides the raw description for points in the point group. Under Property, select the check box. Under Override, either click ⊞ and enter the raw description override value, click ⊞ to attach an external data reference, or click ⊞ then click in the row to specify a user-defined property from the list. For more information, see Using External Data References (page 508) and User-Defined Property Classifications (page 416).

Point Elevation

Overrides the elevation for points in the point group.

Point Style

Overrides the point style for points in the point group. Under Property, select the check box. Click in the Override column to open the Point Style dialog box where you can specify a style.

**NOTE** Changing the override point style updates the point style on the Information tab.

Point Label Style

Overrides the point label style for points in the point group. Under Property, select the check box. Click in the Override column to open the Point Label Style dialog box where you can specify a style.

**NOTE** Changing the override point label style updates the point label style on the Information tab.

**Related procedures:**

■ Creating Point Groups (page 529)

## Point List Tab (Point Group Properties Dialog Box)

This tab displays a read only list of the points in the point group.

Change the order in which the headings are displayed by dragging a heading left or right. Adjust the width of a column by dragging the separator between two column headings left or right.

**Related procedures:**

■ Creating Point Groups (page 529)

## Summary Tab (Point Group Properties Dialog Box)

Use this tab to display information about the point groups properties.

Some information on this tab can be copied to a file for editing or printing using standard Windows cut-and-paste techniques.

Property

Lists point group information by category. Expand or collapse the categories to view the information.

Value

Lists the value for each property.

Query Statement

Lists the point group query. Use the Query Builder tab to modify the query.

**Related procedures:**

■ Creating Point Groups (page 529)

# Point Style Dialog Box

This dialog box defines a point style, which controls the way a point symbol is displayed in a drawing.

## Information Tab (Point Style Dialog Box)

Use this tab to change the point style name and description information, and to review details, such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■ Creating a Point Style (page 420)

# Marker Tab (Point Style Dialog Box)

Use this tab to specify the appearance of the point symbol in the drawing.

## Use AutoCAD Point For Marker

Use AutoCAD Point For Marker

Displays the point using the current AutoCAD point symbol, which is specified by the AutoCAD PDMODE and PDSIZE system variables.

## Use Custom Marker

Use Custom Marker

Displays the point using the specified symbol.

Custom Marker Style

Specifies the symbol to be displayed. Click one of the five symbols on the left to use as a base symbol. Optionally, click the sixth symbol, the seventh symbol, or both, to superimpose over the base symbol. The specified symbol combination is displayed in the Preview window.

## Use AutoCAD Block Symbol For Marker

Use AutoCAD Block Symbol For Marker

Displays the point using a reference to the selected AutoCAD block. The block is scaled using the options specified under Size.
Lists available block definitions in the drawing. Do one of the following:

■ Click a block name to specify the block to be used for the point symbol. The block displays in the Preview window.

■ Right-click in the block list window. Click Browse to select a block located in another folder.

The specified block displays in the Preview window.

## Marker Rotation Angle

Marker Rotation Angle

Specifies the rotation angle for the symbol. Applies to all three symbol types (AutoCAD Point, Custom

Marker, and AutoCAD Block). Enter a value or click ⌖ to specify an angle.

Orientation Reference

Specifies the Marker Rotation Angle:

■ World Coordinate System: Indicates that the marker rotation angle is relative to the world coordinate system.

■ Object: Indicates that the marker rotation angle is relative to an object it is attached to.

■ View: Indicates that the marker rotation angle is relative to the current AutoCAD view direction.

### Size

Options

Specifies symbol scaling:

- Use Drawing Scale: Determines the size of the marker by multiplying the specified value by the drawing scale. Enter the scale factor.

- Use Fixed Scale: Activates the Fixed Scale options.

- Use Size in Absolute Units: The marker size is an absolute value based on the displayed units. Enter the value.

- Use Size Relative to Screen: The size of the marker is a percentage of the drawing screen size. Enter the percentage.

Fixed Scale

Specifies independent fixed scale values when Options is set to Use Fixed Scale. Enter values for X, Y, or Z.

### Preview

Preview

Displays a preview of the specified symbol. Right-click in the Preview window to access commands that you can use to change the Preview window display.

**Related procedures:**

- Creating a Point Style (page 420)

# 3D Geometry Tab (Point Style Dialog Box)

Use this tab to specify the appearance of the point symbol in 3D views.

### 3D Geometry

Point Display Mode

Specifies how the point symbol will be displayed in 3D views:

- Use Point Elevation: Displays the point using its actual elevation value.

- Flatten Points To Elevation: Specifies the elevation that the point is projected (flattened) to. Enter a value for Point Elevation.

- Exaggerate Points By Scale Factor: Exaggerates the elevation of the point by a specified scale factor. Enter a value for Scale Factor.

Point Elevation

Specifies the elevation to which a point is projected when Point Display Mode is set to Flatten Points To Elevation.

Scale Factor

Specifies the scale factor when Point Display Mode is set to Exaggerate Points By Scale Factor.

**Related procedures:**

- Creating a Point Style (page 420)

## Display Tab (Point Style Dialog Box)

Use this tab to change the display and visibility of point components.

View Direction

Specify the context for which you are setting the point display attributes:

■ Plan – the printed drawing

■ Model – the on-screen design environment

■ Profile – projection into a profile view (marker only)

■ Section – projection into a section view (marker only)

Point components include:

■ Marker: The symbol displayed at the point location.

■ Label: The point label, which is specified by the point label style.

For more information, see Display Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■ Creating a Point Style (page 420)

## Summary Tab (Point Style Dialog Box)

Use this tab to display information about a particular point style. You can copy and paste this information to the clipboard.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

**Related procedures:**

■ Creating a Point Style (page 420)

# Create Points Dialog Box

Use this dialog box to create points using a variety of options, including importing point data from a file.

You can expand the Create Points dialog box to display and edit the point creations settings. For more information, see Point Settings (page 410) and Edit Feature Settings - Point Dialog Box (page 2103)

**NOTE** If you specify Save Command Changes To Settings at the Drawing Settings, and enter a different value for Save Command Changes To Settings at the feature level or the command level, an icon displays in the Child Override column of the Drawing Settings to indicate a setting change at a lower level. For more information, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856).

≫ Expand

Expands the Create Points dialog box to display point settings.

≪ Collapse

Collapses the Create Points dialog box so that only the command lists display.

Point Creation Lists

Specifies point creation commands. Select a point creation command from the following lists:

- Miscellaneous: For more information, see Creating Points Using Miscellaneous Methods (page 428).

- Intersection: For more information, see Creating Points at Intersections (page 440).

- Alignment: For more information, see Creating Points Based on Horizontal Alignments (page 459).

- Surface: For more information, see Creating Points Based on Surface Elevations (page 469).

- Interpolation: For more information, see Creating Points by Interpolation (page 474).

- Slope: For more information, see Creating Points Based on Slope (page 483).

- Import Points Category: For more information, see Creating Points by Importing Point Data (page 487).

**Related procedures:**

- Creating Points (page 427)

# Point Groups Dialog Box

Use this dialog box to change the display order for point groups in a drawing and to update out-of-date point groups.

Also, use this dialog box to select a point group when needed, for example to add a point group to a surface definition or to specify a point group when creating a point table.

A point group is out-of-date when  is displayed next to a point group name.

 Show Differences

Displays the Point Group Changes (page 2119) dialog box.

 Update Point Groups

Updates all out-of-date point groups.

Name

Lists the point groups in the order they are drawn. Select a point group to change its position in the display order or to add it to a surface definition. For more information about point group display order, see Changing the Point Group Display Order (page 526).

Description

Lists point group descriptions.

 Up Arrow

Moves the selected point group up in the display order.

 Down Arrow

Moves the selected point group down in the display order.

# Point Group Changes Dialog Box

Use this dialog box to review and update out-of-date point groups.

![icon] Update Point Groups

Updates all out-of-date point groups by applying the changes specified in the Name, Add/Remove, and List columns.

Name

Lists the names of the out-of-date point groups.

Add/Remove

Specifies which of the following actions must be taken to bring the point group up to date:

■ Add: Points in the List column must be added to the point group to bring it up to date.

■ Remove: Points in the List column must be removed from the point group to bring it up to date.

List

Lists the point numbers that must be added to or removed from the point group to bring it up to date.

# Point File Formats Dialog Box

Use this dialog box to manage point import/export formats.

Format List

Specifies the name of the format you want to work with. Click a point file format to select it.

New

Creates a new format.

Copy

Creates a copy of the selected format.

Modify

Edits the selected format. This option is active only when the selected format can be changed.

View

Displays the properties of the selected format. This option is active only when the selected format cannot be changed.

Delete

Deletes the selected format.

# Point File Formats - Select Format Type Dialog Box

Use this dialog box to specify the type of point file format to create.

User Point File

Creates a point file format that describes the contents of an ASCII (text) point data file.

User Point Database

Creates a point file format that describes the content of a Microsoft® Access (*.mdb*) point data file.

**Related procedures:**

■ Understanding Point File Formats (page 496)

# Point File Format Dialog Box

Use this dialog box to view or edit the properties of a user point file format, which is used to import points from or export points to an ASCII (text) file.

### File Information

Format Name

Specifies the name of the point file format. This name appears in the Point File Formats list in the Settings tree. Enter a name.

Default File Extension

Specifies the default file extension of the point data file when you import or export points using this format.
Enter a file extension or select one of the following:

■ .auf: Autodesk Uploadable File, comma delimited. Required values in the file are Point Number, Northing, Easting, Elevation, and Description (in that order).

■ .csv: Comma Separated Value file; ASCII (text) file comma-delimited.

■ .nez: Northing, Easting, and Elevation data

■ .pnt: Point file.

■ .prn: Formatted text, space delimited.

■ .txt: Delimited ASCII (text) file.

■ .xyz: Coordinates *X*, *Y*, and *Z*.

Comment Tag

Specifies how descriptive text is designated in the point data file. Enter the symbol that precedes a comment in the point data file. For example, if you specify # for the comment tag, any line in the point data file that begins with #, such as #Autodesk Point Data, is ignored during import.
The comment tag indicates where the comment begins. A comment always ends at the end of the line.

### Coordinate Zone Transform

For more information, see Point File Format Coordinate Zone Transformation Property (page 497).

Coordinate Zone Transform

Indicates that the format references a coordinate zone.

Select Zone

Opens the Select Coordinate Zone dialog box.

### Format Options

For more information, see Point File Formatting Options (page 498).

Columnated
Specifies that the point data in the point data file is arranged in columns.

Delimited By
Specifies that the point data in the file is separated by the indicated character. Enter the character.

Read No More Than
Specifies the maximum number of points that can be read from a file (during import) or written to a file (during export), beginning at the top of the file. The limit does not include comment lines or errors in the text file. If you specify a limit of 100, then 100 points are imported or exported.

Sample Every
Samples point data at a specified interval. For example, if you specify a sample interval of 100, then every 100th point is imported or exported.

### Format Column Names

Column Headings or <unused>
Opens the Point File Formats - Select Column Name (page 2122) dialog box. Click an <unused> column heading to assign a column name to the column.

**NOTE** Drag a column heading horizontally to quickly change the column order.

### Reference File Column Names

Column Headings
Displays a point data file so that you can reference its contents as you build the format. (See Load and Parse.)

### Options

Load
Opens the Select Source File dialog box where you can load and view a point data file for reference.

Parse
Displays the loaded file in the dialog box using the point file format you are creating.

**Related procedures:**

■ Creating Point File Formats (page 500)


# User Point Database Format Dialog Box

Use this dialog box to view or edit the properties of a user point database format, which is used to import points from or export points to a Microsoft® Access database *(.mdb)* file.

### File Information

Format Name
Specifies the name of the user point database format. This name appears in the Point File Formats list in the Settings tree. Enter a name.

Table Name
Specifies which table within the Microsoft Access file contains point data. Used when a Microsoft Access point database contains multiple tables.

In the Table Name list, click the table in the Microsoft Access file that you want to use. If you do not see any table names listed, then click the Load button to load the *.mdb* file or enter the name of the table you want to use.

### Coordinate Zone Transform

For more information, see Point File Format Coordinate Zone Transformation Property (page 497).

Coordinate Zone Transform

Indicates that the format references a coordinate zone.

Select Zone

Opens the Select Coordinate Zone dialog box.

### Format Column Names

Column Headings or <unused>

Opens the Point File Formats - Select Column Name (page 2122) dialog box. Click an <unused> column heading to assign a column name to the column.

**NOTE** Drag a column heading horizontally to quickly change the column order.

### Options

Load

Opens the Select Source File dialog box where you can load and view a Microsoft® Access file for reference.

**Related procedures:**

■ Creating a User Point Database Format (page 501)

# Point File Formats - Select Column Name Dialog Box

Use this dialog box to specify a column to be included in a point file format.

After you chose a column name, the dialog box displays additional edit boxes that are consistent with the type of data the column contains.

Column Name

Specifies the name of the column to be included in the point file format. For more information, see Point File Format Column Name Property (page 497).

■ <unused>: Designates a column in the point data file that is empty or that contains data you do not want imported or exported.

■ Easting

■ Northing

■ Point Elevation

**NOTE** The elevation value in a point data file can be adjusted during import or export. For more information, see Adjusting Elevation During Import and Transfer (page 499).

■ Point Number

■ Name

■ Raw Description

■ Full Description

**NOTE** You cannot import the full description; you can only export it.

- Grid Northing

- Grid Easting

- Longitude

- Degrees-Longitude

- Minutes-Longitude

- Seconds-Longitude

- Hemisphere-Longitude

- DECDEG Longitude

- DASHED Longitude

**NOTE** A DASHED longitude value is expressed with dashes, such as 73-13-12.67.

- Latitude

- Degrees-Latitude

- Minutes-Latitude

- Seconds-Latitude

- Hemisphere-Latitude

- DECDEG Latitude

- DASHED Latitude

**NOTE** A DASHED latitude value is expressed with dashes, such as 73-13-12.67.

- Thickness: For more information, see Adjusting Elevation During Import and Transfer (page 499).

- User Defined: Specifies a custom column format. Create as many custom columns as you need. Specify a name for the column, a data type, and an invalid indicator value. You should use this option when you need to copy columns from the source file to the destination file during a Transfer Points command .

- Z+: Specifies a positive elevation adjustment. For more information, see Adjusting Elevation During Import and Transfer (page 499).

- Z-: Specifies a negative elevation adjustment. For more information, see Adjusting Elevation During Import and Transfer (page 499).

- Convergence: Contains calculated convergence angles during point export. For more information, see Calculating Convergence Angles During Export (page 499).

- Scale Factor: Contains calculated scale factors for export. For more information, see Calculating Scale Factors During Export (page 499).

- Intensity: Specifies the color intensity of points in a point cloud.

- **RGB-Red**: Specifies the value of red in the RGB color space for points in a point cloud.

- RGB-Green: Specifies the value of green in the RGB color space for points in a point cloud.

- RGB-Blue: Specifies the value of blue in the RGB color space for points in a point cloud.

- Classification: Specifies the LiDAR LAS format Classification value, such as Bare Earth, Low/Medium/High Vegetation, or Infrastructure. For details, see The American Society Photogrammetry and Remote Sensing (ASPRS) LAS Specification.

- \<user-defined\>: Displays the name of a user-defined property. If you are importing point data from and ASCII file that contains user defined columns, you must create a user defined classification for each column. For more information, see Importing Point Data from an ASCII File Containing User Defined Columns (page 504).

**Invalid Indicator**

(Displayed only after you click a column name.) Specifies a value that causes this column to be ignored when detected in the point data file. Enter a value, for example, 999999.

**Precision**

(Displayed only after you click a column name that requires a decimal value.) Enter the number of decimal precision places for the column, up to 12.

**Thickness Name**

(Displayed only after you click the Thickness column name.) Enter a name for the thickness.

**User Defined Column Name**

(Displayed only after you click the User Defined column name.) Enter a heading name for the user-defined column.

**Data Type**

(Displayed only after you click the User Defined column name.) Select the type of data the column contains:

- Double: Contains a double precision floating point value. Use Double for numbers that contain decimal points, including elevations, northings, eastings, grid northings, grid eastings, latitudes, and longitudes.

- Long: Contains a long integer value. Use Long for point numbers.

- String: Contains an alphanumeric value. Use String for point descriptions and point names.

**Related procedures:**

- Creating Point File Formats (page 500)

# Import Points Dialog Box

Use this dialog box to specify options before you import points from an ASCII (text) file or a Microsoft® Access database (*.mdb*) file.

**Format**

Specifies the format determining how the point data is arranged in the file from which the points will be imported.

 Point Format

Opens the Point File Formats (page 2120) dialog box. Select a point file format, edit a point file format, or create a new point file format to use when importing the point data.

**Source File**

Specifies the name of the point data file that contains the points to be imported. Enter a file name, including the full path name.

⊕ File Selector

Opens the Select Source File dialog box. Browse to the folder where the point data file to be imported is located. Select the file name, and click Open.

## Add Points to Point Group

Add Points to Point Group

Specifies whether the imported points are added to a point group:

- Selected: Adds the imported points to the specified point group and to the _All Points point group.

- Cleared: Adds the imported points only to the _All Points point group.

Add Points To Point Group

Specifies the point group to which imported points are added when is selected.

⟨⊕⟩

Opens the Point File Formats - Create Group (page 2125)dialog box. Specify a point group name.

## Advanced Options

Do Elevation Adjustment If Possible

Specifies that elevation adjustments are performed during import. The point file format must contain Z+, Z-, or Thickness columns. For more information, see Adjusting Elevation During Import and Transfer (page 499).

Do Coordinate Transformation If Possible

Specifies that coordinate transformations should occur during import. The point file format must have a coordinate zone assigned to it, and the current drawing must have coordinate zone and transformation information defined. The points that are imported are transformed to match the zone of the current drawing.

Do Coordinate Data Expansion If Possible

Specifies that coordinate data properties of the points, such as degrees, minutes, seconds, and hemisphere for latitude and longitude, should be calculated if possible. These values are calculated from known coordinate data information contained in the point data file, such as grid northing and grid easting.

**Related procedures:**

- Importing Point Data (page 503)


# Point File Formats - Create Group Dialog Box

Use this dialog box while importing points to either create a point group that includes all the imported points or to add the imported points to an existing point group.

Point Group

Specifies the name of the point group to which the points are added. If the point group does not exist, it is created.

**Related procedures:**

- Creating Point Groups (page 529)

# Export Points Dialog Box

Use this dialog box to specify options before you export points to an ASCII (text) file or a Microsoft Access database (*.mdb*) file.

Format

Specifies the format describing how the point data will be arranged in the exported file.

⊞ Point Format

Opens the Point File Formats (page 2120) dialog box. Select a point file format, edit a point file format, or create a new point file format to use when importing the point data.

Destination File

Specifies the name of the file to which the exported points will be written. Enter a file name, including the full path name.

⊞ File Selector

Opens the Select Destination File dialog box. Browse to the folder where the exported point data file will be located. Enter a file name, and click Save.

## Limit to Points in Point Group

Limit To Points In Point Group

Specifies whether or not a point group is used to specify the points to be exported:

■ Selected: Only the points in the specified point group are exported.

■ Cleared: All points are exported.

Point Group

Specifies the name of the point group that contains the points to be exported when Limit To Points In Point Group is selected.

## Advanced Options

Do Coordinate Transformation If Possible

Specifies that coordinate transformations should occur during export. The point file format must have a coordinate zone assigned to it and the drawing you are exporting points from must have coordinate zone and transformation information defined. The points that are exported are transformed to match the zone assigned to the point file format.

Do Coordinate Data Expansion If Possible

Specifies that missing coordinate data properties of the points being exported, such as degrees, minutes, seconds, and hemisphere for latitude and longitude, are calculated if possible. These values are calculated from known coordinate data information, such as grid northing and grid easting.

**Related procedures:**

■ Exporting Point Data (page 505)

# Transfer Points Dialog Box

Use this dialog box to specify options before you transfer points from a source point data file (either text or *.mdb*) to a destination point data file (either text or *.mdb*).

**Source**

Format

    Specifies the format describing how the point data is arranged in the source file.

➕ Source

    Specifies the name of the point data file from which the point data is read. Enter a file name, including the full path name.

File Selector

    Opens the Select Source File dialog box. Browse to the folder where the source point data file is located. Select the file name, and click Open.

**Destination**

Format

    Specifies the format describing how the point data will be arranged in the destination file.

Destination

    Specifies the name of the file to which the point data is written. Enter a file name, including the full path name.

🔍 File Selector

    Opens the Select Destination File dialog box. Browse to the folder where the point data file will be located. Enter a file name, and click Save.

**Advanced Options**

Do Elevation Adjustment If Possible

    Specifies that elevation adjustments are performed during import. The format must contain Z+, Z-, or Thickness columns. For more information, see Adjusting Elevation During Import and Transfer (page 499).

Do Coordinate Transformation If Possible

    Specifies that coordinate transformations should occur during transfer. Both source and destination point file formats must have coordinate zones assigned to them. The point data in the source file is transformed during transfer to match the zone specified in the destination file point file format.

Do Coordinate Data Expansion If Possible

    Specifies that missing coordinate data properties of the points in the source file, such as degrees, minutes, seconds, and hemisphere for latitude and longitude, should be calculated, if possible, and transferred to the destination file. These values are calculated from known coordinate data information such as grid northing and grid easting in the source file.

**Additional Options**

Manage

    Opens the Point File Formats (page 2119) dialog box. Create, copy, or change a format.

**Related procedures:**

■   Transferring and Converting Point Data (page 506)

# Description Key Set Dialog Box

Use this dialog box to specify the name and description of a description key set.

Name

    Specifies the name of the description key set. Enter a name.

Description

Specifies a description for the description key set. Enter a description.

**Related procedures:**

■ Creating a New Description Key Set (page 547)

# Description Key Sets Search Order Dialog Box

Use this dialog box to specify the order in which description key sets are searched during description key matching.

The description key sets are listed in the order in which they are searched. The description key set at the top of the list is searched first.

Name

Lists the name of the description key set. Click the name to select the description key set.

Description

Lists the description of the description key set.

⬆ Up Arrow

Moves the selected description key set up in the search order.

⬇ Down Arrow

Moves the selected description key set down in the search order.

**Related procedures:**

■ Changing the Description Key Sets Search Order (page 548)

# Description Key Editor

Use this dialog box to edit description key properties.

You can also edit description keys in the Prospector item view. However, because the DescKey Editor is a separate window, it provides more viewing area and greater flexibility.

**Code and Format**

These properties are required in every description key.

Code

Specifies what points match this description key during description key matching. Click in the cell and then enter a code. For more information, see Description Key Code (page 540).

Format

Specifies the format used to translate the raw description into the full description. Click in the cell and enter a format. Enter $* if you want the full description to be the same as the raw description. For more information, see Description Key Format (page 542).

**Styles and Layer**

For more information about how point style, point label style, and point layer are used to display a point, see Controlling the Appearance of Points in a Drawing (page 425).

Point Style

Specifies the point style to be referenced by a point that is created using the description key.

To specify a point style:

- Select the Point Style check box.

- Click in the cell to open the Point Style (page 2114) dialog box. Select a point style.

---

**NOTE** Scale and rotation values in the point style are overridden by description key scale and rotation overrides.

---

Point Label Style

Specifies the point label style to be referenced by a point that is created using the description key.
To specify a point label style:

- Select the Point Label Style check box.

- Click in the cell to open the Point Label Style (page 1805) dialog box. Select a point label style.

Layer

Specifies the layer for the point.
To specify a layer:

- Select the Layer check box.

- Click in the cell to open the Layer Selection dialog box. Select a layer.

## Scale Override Properties

Use the Scale Override properties of a description key to specify how the point symbol is scaled when a point is created using the description key.

Scale values specified in this part of the description key take precedence over any scale values specified in the point style referenced by the description key. If scale parameters have been set, hold your cursor over

□ in the Code column, to display a tooltip with the scale parameters and values.

Select Apply To X-Y or Apply To Z before you specify a scale using Scale Parameter, Fixed Scale Factor, or Use Drawing Scale properties.

Scale Parameter

Specifies the position (1-9) in the raw description of the parameter that contains the value used to scale the point symbol when the description key is matched.
To specify a scale parameter:

- Select the Scale Parameter check box.

- Click in the cell and select a parameter.

For more information, see Specifying Point Symbol Scaling and Rotation Using Description Key Parameters (page 544).

Fixed Scale Factor

Specifies a fixed scale, used to scale the point symbol when the description key is matched.
To specify a fixed scale factor:

- Select the Fixed Scale Factor check box.

- Click in the cell and enter a scale factor.

Use Drawing Scale

Specifies that the drawing scale is used to scale the point symbol when the description key is matched.
Click in the cell then select Yes to use the drawing scale.

Apply To X-Y

Specifies that description key scaling is applied to the *X-Y* axis of a point when the description key is matched. Click in the cell then select Yes to specify *X-Y* scaling.

Apply To Z

Specifies that description key scaling is applied to the Z axis of a point when the description key is matched. Click in the cell then select Yes to specify Z scaling.

### Rotation Override Properties

Use the Rotation properties of a description key to specify how the point symbol is rotated when a point is created using the description key.

Rotation values specified in this part of the description key take precedence over any rotation values specified in the point style that is referenced by the description key.

If rotation parameters have been set, hold your cursor over in the Code column to display a tooltip with the rotation parameters and values.

Rotate Parameter

Specifies the position (1-9) in the raw description of the parameter containing the value used to rotate the point symbol when the description key is matched.
To specify a rotation parameter:

■ Select the Rotation Parameter check box.

■ Click in the cell then select a parameter.

For more information, see Specifying Point Symbol Scaling and Rotation Using Description Key Parameters (page 544).

Fixed Rotation

Specifies a fixed rotation used to rotate the point symbol when the description key is matched.
To specify a fixed rotation factor:

■ Select the Fixed Rotation check box.

■ Click in the cell and enter a rotation value.

Rotation Direction

Specifies whether the values specified for the Rotate Parameter property or the Fixed Rotation property are clockwise angles or counterclockwise angles.

### Related procedures:

■ Editing Description Keys (page 556)

## Point Editor

Use the window to edit drawing point properties.

Each row in the Point Editor contains the properties for a single point. To edit a point property value, click in the cell containing the property to activate it. If a cell is shaded, you cannot change the value.

**NOTE** You can also edit drawing points in the Prospector item view. However, because the Point Editor is a separate window, it provides more viewing area and greater flexibility.

These drawing point properties are displayed in the Point Editor and in the drawing Points collection list view:

**Point Number**

Specifies the point number. Point numbers must be unique. Enter an integer.

**Easting**

Specifies a local easting value for the point. If the transformation settings are enabled for the drawing when you edit this value, the corresponding values for grid easting and latitude are automatically updated. Enter a value with up to 12 digits of precision.

**Northing**

Specifies a local northing value for the point. If the transformation settings are enabled for the drawing when you edit this value, the corresponding values for grid northing and longitude are automatically updated. Enter a northing value with up to 12 digits of precision.

**Point Elevation**

Specifies an elevation for the point. Enter an elevation value.

**Name**

Specifies an optional point name, which must be unique within a drawing or a project.

**NOTE** Point names are not case sensitive.

**Raw Description**

Specifies a raw description for the point, which is often the description entered by the surveyor in the field. Enter alphanumeric characters. Any character, including a blank, can be used.

**Full Description**

Specifies an expanded description that is created from the raw description using the description format. If a point does not have a description format, the full description is the same as the raw description. You cannot edit this field directly. The value can be set initially using description key matching. For more information, see Description Keys (page 537). You can update the full description of a single point either by changing the points description format or by changing the points raw description. For more information, see Description Key Format (page 542).

**Description Format**

Specifies a format that translates the points raw description into a full description. If a point does not have a description format, the full description is the same as the raw description. For more information, see Description Key Format (page 542).

**Grid Easting  Grid Northing**

Displays the calculated grid easting and grid northing values for the point, relative to the coordinate zone and the transformation settings specified for the drawing.

**Longitude  Latitude**

Displays the latitude and longitude for a point, relative to the coordinate zone and the transformation settings specified for the drawing.

**Scale Factor**

Displays the scale factor value of a point, relative to the coordinate zone and the transformation settings specified for the drawing.

**Convergence**

Displays the convergence value of a point, relative to the coordinate zone and the transformation settings specified for the drawing.

**Point Style**

Specifies the point style for the point. Click in the cell to specify a new point style or remove a point style.

If this field is empty, the point displays using a point style specified by a point group. Also, the point style specified in this field might not be used to display the point in the drawing. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Point Label Style**

Specifies the point label style for the point. Click in a cell to specify a new point label style or remove the point label style.

If this field is empty, the point uses a point label style specified by a point group. Also, the point label style specified in this field might not be used to display the point in the drawing. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Point Layer**

Specifies the layer for the point. Click in the cell to display the Layer Selection dialog box.

If this field is empty, the point displays using a layer specified by a point group. For more information, see Controlling the Appearance of Points in a Drawing (page 425).

**Project Version**

For project points, specifies the version number of the local copy of the point in the drawing.

**X-Y Scale**

Specifies the X-Y scale factor for the point symbol. This value can be set initially using description key matching. For more information, see Description Keys (page 537).

**Z Scale**

Specifies the Z scale factor for the point symbol. This value can be set initially using description key matching. For more information, see Description Keys (page 537).

**Rotation**

Specifies the rotation value for the point symbol. This value can be set initially using description key

matching. For more information, see Description Keys (page 537). Enter a value or click ⬉ and pick two points in the drawing to specify the rotation angle.

These point properties are displayed in a project Points collection list view for a project point:

**Point Number, Easting, Northing, Point Elevation, Name, Raw Description**

See the preceding text for descriptions of these properties.

**Version**

Displays the most recent project version for the point.

**Modified By**

Displays the login name of the person who last modified the point.

**Modified Date**

Displays the date when the point was last modified.

**Current State**

Displays the current state of the point with respect to the project.

**Current State By**

Displays the project member who set the points current state.

**Current State Date**

Displays the date the state of the point was last changed.

**Checked-Out Drawing**

If the point is checked out, displays the drawing it is checked out to.

Checked-Out Host

    If the point is checked out, displays the computer it is checked out to.

**Related procedures:**

■   Editing Points Using the Point Editor (page 488)

# Geodetic Calculator Dialog Box

Use this dialog box to calculate geodetic information relative to the zone and the transformation values specified in the drawing settings.

To use the calculator, enter values for known coordinate properties, and the remaining coordinate values are calculated.

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**NOTE** If transformation settings for a drawing are not enabled, the Select Point, Local Northing, Local Easting, and Local Elevation options are not active.

---

Specify Point

    Specifies local northing and easting values using a drawing point. Click a point in the drawing.

Create Point

    Creates a point with the specifications you enter in the value column. After you enter or edit values, you can click this icon to create a new point. At the command line you are prompted to enter a point description and elevation.

Zone Description

    Displays read-only information about the zone specified in the drawing.

Point Number

    Specifies a starting point number from which local northing and easting values are read. Grid Northing, Grid Easting, Latitude, and Longitude are calculated using the drawing zone. Enter a point number.

Latitude

    Specifies latitude. Enter a latitude value using the format specified for the Lat Long drawing ambient setting.
    Changing this property updates the fields for Grid Northing, Grid Easting, Local Northing, Local Easting, Scale Factor, and Convergence.

Longitude

    Specifies longitude. Enter a longitude value using the format specified for the Lat Long drawing ambient setting.
    Changing this property updates the fields for Grid Northing, Grid Easting, Local Northing, Local Easting, Scale Factor, and Convergence.

Grid Northing

    Specifies grid northing. Enter a grid northing value.
    Changing this property updates the fields for Latitude, Longitude, Local Northing, Local Easting, Scale Factor, and Convergence.

Grid Easting

    Specifies grid easting. Enter a grid easting value.
    Changing this property updates the fields for Latitude, Longitude, Local Northing, Local Easting, Scale Factor, and Convergence.

Local Northing

Specifies local northing. Enter a local northing value.
Changing this property updates the fields for Latitude, Longitude, Grid Northing, Grid Easting, Scale Factor, and Convergence fields to be updated.

Local Easting

Specifies local easting. Enter a local easting value.
Changing this property updates the fields for Latitude, Longitude, Grid Northing, Grid Easting, Scale Factor, and Convergence.

Local Elevation

Specifies the local elevation for the specified point number.

Scale Factor

Displays the scale factor for the specified coordinate values. This read-only value is updated when coordinates are recalculated.

Convergence

Displays the convergence for the specified coordinate values using the format specified for the Lat Long drawing ambient setting. This read-only value is updated when coordinates are recalculated.

Sea Level Corrections Applied

Specifies whether sea level corrections are applied in Drawing Settings.

Grid Scale Factor Applied

Specifies whether the grid scale factor is applied in Drawing Settings.

**Related procedures:**

■  Geodetic Calculator (page 515)


# Duplicate Point Number Dialog Box

Use this dialog box to handle point number conflicts arising from trying to create a point with the same point number as an existing point.

In the following descriptions, source point is the point that the command is attempting to create, and destination point is the existing point.

### Duplicate Point Number Resolution

IMPORTANT  For duplicate Survey point numbers, Overwrite and Ignore are the only Resolution options. Selecting the Ignore option leaves the COGO point values intact. Options for Add An Offset From and Sequence From are not available.

Resolution

Specifies how the point number conflict is resolved:

■  Add An Offset: Adds the offset specified for the Add An Offset From option to the source point number. For example, if you enter an offset of 200, source points with numbers 1, 2, and 3 are re-numbered 201, 202, and 203.

■  Merge: Overwrites point data that exists in the destination point with data that exists in the source point and preserves data in the destination point that is not supplied by the source point.

■  Overwrite: Overwrites the destination point.

■  Sequence From: Begins renumbering the source points using the point number specified in the Sequence From option.

■ Use Next Point Number: Assigns the next unused point number to the point.

Add An Offset From

Specifies the offset to be added to the source point number when the Resolution is set to Add An Offset. Enter an integer.

Sequence From

Specifies the starting sequence number for renumbering the source points when the Resolution is set to Sequence From. Enter an integer.

Apply To All Duplicate Point Numbers

Specifies that the Resolution setting is applied to any additional duplicate point numbers encountered by the command. If a duplicate point number occurs that cannot be resolved based on the Resolution setting, this dialog box displays again.

# Duplicate Point Name Dialog Box

Use this dialog box to handle point name conflicts arising from trying to create a point with the same point name as an existing point.

In the following descriptions, source point is the point that the command is attempting to create, and destination point is the existing point.

### Duplicate Point Name Resolution

Resolution

Specifies how the point name conflict is resolved:

■ Counter: adds a numerical suffix to the current point name. A new point is created. The second instance of the duplicate point name would be <point name> (1).

■ Specify: Displays the Point Name edit box to specify a name for the imported point.

■ Use Name Template: Creates a new point and assigns the point name based on the point name template.

Point Name

Specify a point name if you select Specify as the Resolution.

Apply To All Duplicate Point Numbers

Select to apply the resolution to all duplicate point names. If you select Specify as the Resolution type, this check box is not available.

# Point Table Creation Dialog Box

Use this dialog box to create a point table in a drawing.

Most of the controls in this dialog box are generic table creation controls. For a description of these controls, see Table Creation Dialog Box (page 2441).

Use the following controls to specify the points to be included in the point table:

⊡ Point Group Selector

Displays the Point Groups (page 2118) dialog box. Specifies a point group. The points in the point groups point list are added to the point table.

▣ Point Selector

Selects the points to be added to the point table.

**Related procedures:**

■ Point Tables (page 422)

# Create External Data Reference Dialog Box

Use this dialog box to create an external data reference (XDRef) or change the properties of an existing XDRef.

Name

Specifies the file name for the XDRef.

Description

Specifies an optional description.

Database

Specifies the name and the path for the external database file *(.mdb file)*.

Password Protected Database

Specifies that a password must be entered if this is selected.

Table

Specifies the column data for the XDRef.

Index Column

Specifies the index column containing the integers that correspond to the point numbers within the point group that you are overriding.

Value Column

Specifies the column that contains the values that override the point group values when you specify this XDRef as the override.

**Related procedures:**

■ Using External Data References (page 508)

# Create Blocks from COGO Points Dialog Box

Use this dialog box to specify a selection of COGO points to use in creating an AutoCAD BLOCK for each Civil point.

**Select COGO Points**

Spatial Filter

Specifies a filter.

■ None: Uses only those points contained in the point group that is specified under Point Groups.

■ Current Display: Uses the current display to define the spatial selection of points in the drawing.

■ Define Window: Uses a crossing window in the drawing to define the spatial selection of points.

Point Groups

Click 🔲 to open the Point Groups dialog box, which lists all point groups in the drawing. Specify the point group(s) you want to include in the block creation.

**Block Output**

Block Creation

■ Use Existing Block: to use this option the block must contain at least three attribute definitions exactly named ELEV, POINT, and DESC. If there are no blocks in the drawing with these exact attributes, this option is not available.

■ Make New Block: specifies a new block and block name. If a block has two or more instances of the same attribute, each instance is filled with the same value.

Block Layer

Specifies the layer the block is placed on.

**Related procedures:**

■ Creating Blocks from COGO Points (page 513)

# Convert Land Desktop Points Dialog Box

Use this dialog box to enter the point settings for the Land Desktop points you are converting.

For a description of the point settings, see Edit Feature Settings - Point Dialog Box (page 2103).

# Profiles Dialog Boxes

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The following topics provide information about the AutoCAD Civil Profiles dialog boxes.

## Band Set Dialog Box

Use this dialog box to create a set of data band styles for profile views.

Use a band set to apply the same set of data bands to many profile views.

**See also:**

- Profile View Band Styles (page 1038)
- Editing Profile Views (page 1113)

## Information Tab (Band Set Dialog Box)

Use this tab to change the name and description for a band set, and to review details such as latest modification date.

Name
  Specifies the name of the band set.

Description
  Specifies a description of the band set.

Created By
  Displays the Windows login name of the person who created the band set.

Date Created
  Displays the date and time the band set was created.

Last Modified By
  Displays the Windows login name of the person who last modified the band set.

Date Modified
  Displays the date and time the band set was last modified.

# Bands Tab (Band Set Dialog Box)

Use this tab to specify which data bands are included in the set, along with their styles and positions relative to the profile view grid.

Band Type

Specifies which type of data band to add to the set: Profile Data, Vertical Geometry, Horizontal Geometry, Superelevation, Sectional Data, or Pipe Data.

Select Band Style

Specifies a style for the band. To edit a style or create a new one, click  and select from the list of operations. For more information, see Select Style dialog box (page 1805). To examine the details of an existing style, select the style by name in the list and click . For more information, see Style Detail dialog box (page 1804).

Add >>

Adds the specified data band type to the style set. Before clicking this button, ensure that the settings are correct for data band type, style, and location.

## List of Bands

Location

Specifies either the top or bottom of the profile view. The table below this field shows the current location of the bands in the current band style set.

Band Type

Specifies the data band type.

Style

Specifies the style for the band type.

Description

Specifies optional description of the band type

Gap

Specifies the distance between the current data band and the adjacent profile view grid or data band. For bands below the profile view grid, the gap is measured from the top of the current band to the bottom of the band or grid above it. For bands above the profile view grid, the gap is measured from the bottom of the current band to the top of the band or grid below it.

Geometry Points

For Profile Data bands, specifies the alignment and profile geometry points to label in the data band. Click  to open the Geometry Points to Label in Band dialog box (page 2162).

Label Start Station

Specifies whether the start end of the band is labeled.

Label End Station

Specifies whether the end of the band is labeled.

Label Origin

Specifies whether the profile view origin is labeled with its station and elevation values.

Weeding

Used only for labels at vertical geometry points on profile data bands. Vertical geometry points or grade breaks that are closer than the weeding factor are removed, making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance.

Stagger Labels

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left.

Stagger Line Height

Specifies the height of the label stagger line.

Band Position Controls

Changes the position of data bands in the set and deletes unwanted bands.

⬆ Moves selected data bands up.

⬇ Moves selected data bands down.

✖ Deletes selected bands.

Match major/minor increments to vertical grid intervals

Specifies if the profile data band major/minor interval distances for profile views matches the profile view style's major/minor grid spacing intervals. Select the checkbox to match major/minor increments to vertical grid intervals. If this checkbox is selected, the major/minor interval fields are unavailable.

# Copy Profile Data Dialog Box

Use this dialog box to copy all or part of an existing profile.

Copying a profile can be useful when you want to design a profile that runs parallel to another profile in the vertical plane. For example, you can create a ditch profile that follows the shape of a centerline, but is at a lower elevation.

## Source Profile Information

Name

Displays the name of the copied profile

Style

Displays a description for the copied profile.

## PVI Range

Select one of the following options:

All

Copies the entire profile.

Station Range

Copies a series of PVIs between two station values. Default values for the station range are the beginning and end of the profile. If a station equation is in effect, use the station equation values to specify a range

## Destination Profile Options

These options determine how the copy is used: either to create a profile or to overwrite an existing profile.

Overwrite Existing Profile

Replaces an existing profile with the specified source profile. If you select this option, use the drop-down list to specify the profile to be overwritten. To copy all styles, descriptions, and other attributes from the source profile to the specified destination profile, select the Overwrite All Properties check box. Otherwise, only the linear data is overwritten.

Create New Profile

Copies the current profile, subject to any limitations specified in the PVI Range section. The name of the new profile is created according to the current naming template, like the name of any new profile. The new profile is superimposed on the source profile in the profile view. To work on one of these profiles without affecting the other, select it by name on the Prospector tab or in a dialog box. Control the visibility of individual profiles in the Profile View Properties dialog box (page 2179)

**Related procedures:**

■ Copying a Profile (page 1057)

# Create Profile Band Style Dialog Box

Click the type of profile view band style you want to create, then click OK.

After selecting the band style, the Profile View Band Style dialog box (page 2164) is displayed for you to create the style.

**Related procedures:**

■ Profile View Band Styles (page 1038)

# Create Profile - Draw New Dialog Box

Use this dialog box to configure the settings for a new layout profile.

After using this dialog box, click OK to open the Profile Layout Tools dialog box (page 2170) and prepare drawing tools. Draw the profile on the profile view grid by selecting PVIs (points of vertical intersection).

**Related procedures:**

■ Creating Layout Profiles (page 1048)

# General Tab (Create Profile - Draw New Dialog Box)

Use this tab to enter basic information about the profile, including style, layer, and label set.

Alignment

Specifies the horizontal alignment along which to create the profile.

Name

Specifies the name of the profile. Each profile must have a unique name.

Description

Specifies an optional description of the profile.

**Profile Style**

Profile Style List

Displays the current style. Click the arrow to select another profile style in the drawing.

Style Selection

Specifies the style options. Create a style, copy or edit the current style selection, or pick a style from drawing.

 Style Detail

Opens the Style Detail dialog box. Preview the style and creation information.

### Profile Layer

Profile Layer List

Displays the layer on which the profile object is created.

 Object Layer

Opens the Object Layer dialog box (page 1985), in which you can select or create a layer on which the profile is drawn.

### Profile Label Set

Profile Label Set List

Displays the default profile label set. You can either accept the default label set, or use the list to select a different one from the drawing.

 Label Set Selection

Edits or copies the current label set, or creates a label set. The Style Detail icon previews the current style.

 Style Detail

Opens the Style Detail dialog box. Preview the label set and creation information.

# Design Criteria Tab (Create Profile - Draw New Dialog Box)

Use this tab to specify the profile design criteria settings, including the design criteria file and design check set.

Alignment

Specifies the horizontal alignment along which to create the profile.

Name

Specifies the name of the profile. Each profile must have a unique name.

Description

Specifies an optional description of the profile.

### Use Criteria-Based Design

Specifies whether to apply design criteria to the profile. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

### Use Design Criteria File

Use Design Criteria File

Specifies whether to associate a design criteria file with the profile. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.
For more information, see Profile Standards in the Design Criteria File (page 1045).

Design Criteria File Selector

Specifies the design criteria file to associate with the profile. The design criteria file that is applied to the parent alignment is displayed by default. Click  to select another design criteria file.

Default Criteria Table

Displays the profile standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

### Use Design Check Set

Use Design Check Set Check Box

Specifies whether to associate a design check set with the profile. If this check box is cleared, then the Design Check Set list is not available.

For more information, see Profile Design Check Sets (page 1046).

Design Check Set List

Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

 Design Check Set Selection

Edits or copies the current design check set, or creates a design check set.

# Create Profile from Surface Dialog Box

Use this dialog box to create profiles from existing surfaces, including profiles at offsets to parent alignments.

Each profile created here must be associated with a horizontal alignment and one or more surfaces.

Alignment

Specifies the horizontal alignment along which to create the profile. Select an alignment by name, or click to select an alignment in the drawing.

Select Surfaces

Lists all surfaces in the current drawing. Select surfaces by name, or click  to select surfaces in the drawing. To select multiple surfaces, hold down the Shift key while clicking with the mouse.

Station Range

Specifies the range of stations along the parent alignment that are included in the profile. Default values represent the full length of the alignment. To set a different length, enter station values numerically, or click  to select stations in the drawing.

Sample Offsets

Specifies whether to create offset profiles. Select this option if you want offsets. Enter numeric values for the offset distances. Enter positive numbers for right offsets, negative numbers for left offsets, and commas to separate the values (for example, 10,-10,25). Click Add to add each set of profiles to the profile list.

## Profile List

Lists the profiles that have been created for the specified alignment and surfaces. You cannot edit profiles that were created in a previous session. You can edit Profiles created in the current session.

Name

Specifies the name of the profile. Default name strings are generated as defined in Profile Settings dialog box (page 2156). Existing profiles for the parent alignment are also displayed in this list as unavailable entries.

Description

Specifies optional description of the profile.

Type

Specifies whether the profile represents surface or design layout.

Data Source

For surface profile, displays the name of the surface represented in the profile. For superimposed profiles, displays the name of the superimposed profile. For corridor profiles, specifies the name of the source corridor. Not used for layout profiles.

Offset

Specifies the offset distance from the centerline of the parent horizontal alignment: 0 for a centerline profile, a positive number for a right offset, a negative number for a left offset.

Update Mode

For surface profiles only. Specifies whether the profile is dynamic or static.
**Dynamic**: The profile automatically updates to reflect changes in the surface elevation and the route traced by the parent horizontal alignment.

**Static**: The profile shows surface elevations at the time of its creation. It does not update to reflect changes that occur later.

Layer

Specifies the drawing layer on which the profile is placed.

Style

Specifies the style used by the profile.

Station Start/End

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

Elevation Minimum/Maximum

Specifies the highest and lowest elevation values that occur along the profile.

Remove

Removes a selected profile from the profile list. Any profiles removed in this way are unavailable for drawing in a profile view.

Draw in Profile View

Opens the Create Profile View dialog box (page 2145), configured to draw the current profiles.

OK

Saves the profile data and closes the Create Profile from Surface dialog box. If one or more profile views are created along the alignment, the profiles you created are added to the views.

Cancel

Cancels the creation of profiles.

**Related procedures:**

■ Creating Surface Profiles (page 1047)

■ Understanding the Profile Object (page 1019)


# Create Profile View Wizard

Use this wizard to create single or multiple profile views.

Some controls on this wizard are disabled when it is used with the plan production tools (page 1691).

**Related procedures:**

■ Creating Profile Views (page 1108)

■ Understanding Plan Production Tools (page 1691)

# General Page (Create Profile View Wizard)

Use this page to specify basic information about the profile view, including the parent alignment and the profile view name, description, style, and layer.

**NOTE** The controls on this page are not available when multiple profile views are created as part of the plan production process.

Select Alignment

Specifies the name of the parent horizontal alignment for the profile that appears in the profile view. Select a name from the list or click to pick an alignment from the drawing.

Profile View Name

Specifies an editable, system-generated name. The name must remain unique within the drawing.

Description

Specifies an optional description of the profile.

Profile View Style

Specifies the style for the profile view. Provides standard controls you use to review or change the style.

Profile View Layer

Specifies the drawing layer for the profile view. To review or change layer data, click ⬗ .

Show Offset Profiles By Vertically Stacking Profile Views

Specifies to create stacked profile views (page 2483) views. Selecting this option causes the Stacked Profile page (page 2147) to become available.

# Station Range Page (Create Profile View Wizard)

Use this page to specify the station range to which the profile view is drawn.

**NOTE** This page is disabled when multiple profile views are created as part of the plan production process.

Automatic

Specifies the full station range for the horizontal alignment.

User Specified Range

Limits the station range in the profile view. Either enter numeric station values in the Start and End fields, or click ⬗ to select the range on the alignment in the drawing.

Length of Each View

Specifies a length for each profile view segment in a multiple profile view.

# Profile View Height Page (Create Profile View Wizard)

Use this page to specify the profile view height and any split profile view settings.

**Profile View Height**

Automatic

Specifies the full height of the highest profile. Includes a buffer region above the maximum and below the minimum elevations.

User Specified

Specifies the height to which the profile view is drawn. If a profile extends beyond the user-specified value, it is either split according to the Split Profile View settings or clipped.

Profile View Datum By

Specifies how to place the datum for all profiles within the profile view.

- **Minimum Elevation**—All profile datum values are based on the lowest datum value of all profiles in the profile view. This option works well for profiles that have relatively high variation in elevation.

- **Mean Elevation**—Profile datum values are based on the mean value of all profiles drawn in the profile view. Profiles are drawn in the center of the profile view, which places an equal amount of space above and below the profile in the profile view. This option is useful in profiles that have relatively little variation in elevation.

## Split Profile View

**NOTE** The Split Profile View settings are enabled only if the Profile View Height is set to User Specified.

Split Profile View

Specifies that if the height of the profile extends beyond the User-Specified height value of the profile view, the profile view splits.

First Split View Style

Specifies the profile view style to use for the first split profile view segment. Use the standard controls to edit the style or create a new one.

Intermediate Split View Style

Specifies the profile view style to use for the all split profile view segments between the first and last segments. Use the standard controls to edit the style or create a new one.

Split Station

Specifies the horizontal location to split the profile view: previous major grid, previous minor grid, or exact station.

Datum Option

Specifies the vertical location to split the profile view: previous major grid, previous minor grid, or exact elevation.

Last Split View Style

Specifies the profile view style to use for the last split profile view segment. Use the standard controls to edit the style or create a new one.

# Stacked Profile Page (Create Profile View Wizard)

Use this page to specify the settings for stacked profile views (page 2483) views.

**NOTE** This page is not available if the Show Offset Profiles By Stacking Views check box on the General page (page 2146) is cleared.

Number of Stacked Views

Specifies the total number of profile views to create. This unitless value can be any integer from 2 to 9. The graphic on the right-hand side of the page changes to reflect the number selected.

Gap Between Views

Specifies the amount of spacing (in plotted units) to leave between stacked profile views. A negative value in this field causes the stacked profile views to overlap.

Top View Style
    Specifies the profile view style to use for the top-most profile view.

Middle View Style
    Specifies the profile view style of all profile views between the top-most and bottom-most profile views.

Bottom View Style
    Specifies the profile view style to use for the bottom-most profile view.

Preview (unlabled)
    Displays the number of profile views specified in the Number of Stacked Views field.

# Profile Display Options Page (Create Profile View Wizard)

Use this page to view and change settings for all the profiles associated with the specified alignment.

## Select Stacked View To Specify Options For

**NOTE** This control is not available if the Show Offset Profiles By Stacking Views check box on the General page (page 2146) is cleared.

Displays a list of the stacked profile views. The number of profile views in this list depends on the Number Of Stacked Views specified on the Stacked Profile page (page 2147). Select a profile view from the list, and then use the Specify Profile Display Options table to specify the settings for that stacked profile view.

## Specify Profile Display Options

Change column widths in the table to make it easier to see the contents of cells. Use the following controls to modify profiles in the view.

 Select All Profiles
    Selects the Draw check box for all profiles listed in the table.

 Deselect All Profiles
    Clears the Draw check box for all profiles listed in the table.

Name
    Specifies the name of the profile.

Draw
    By default, specifies that all profiles are drawn in the profile view. To omit a profile from the profile view, clear its check box.

Clip Grid
    Specifies which profile controls the horizontal and vertical extents of the profile view grid. This setting works only if the profile view style Grid Options are set to clip the grid. For further information, see Grid Tab (Profile View Style Dialog Box) (page 2186).

Split At
    Specifies which profile to favor when splitting a profile view containing multiple profiles.
    For example, a profile view often contains both existing ground and finished ground profiles. Selecting Split At for the finished ground profile ensures that split occurs at the finished ground profile elevation.

Description
    Specifies an optional description of the profile.

Type

Specifies whether the profile is a surface profile , layout profile , superimposed profile , or corridor profile .

Data Source

For surface profile, displays the name of the surface represented in the profile. For superimposed profiles, displays the name of the superimposed profile. For corridor profiles, specifies the name of the source corridor. Not used for layout profile.

Offset

Specifies the offset distance from the centerline of the parent horizontal alignment: either 0 for a centerline profile, a positive number for a right offset, a negative number for a left offset.

Update Mode

For surface profiles only, specifies whether the profile is dynamic or static.
**Dynamic**: The profile automatically updates to reflect changes in the surface elevation and the route traced by the parent horizontal alignment.

**Static**: The profile shows surface elevations at the time of its creation. It does not update to reflect changes that occur later.

Layer

Specifies the drawing layer on which the profile is placed.

Style

Specifies the style used by the profile. If you change the style here, it changes the profile properties and the style is retained as the specified style when creating additional profile views. Use the standard controls to edit the style or create a new one.

Override Style

Specifies whether the profile style is overridden (changed) for the current profile view. You can override the style by clicking the check box and selecting a different style. If you do, the style change applies only to the current profile view. It does not change the style in the profile properties.

Labels

Click to open the Pick Profile Label Set dialog box where you can edit the profile labels.

Station Start/End

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

Elevation Low/High

Specifies the highest and lowest elevation values that occur along the profile.

Alignment

Specifies either the parent alignment of current profile view or the parent alignment of any superimposed profiles.

## Pipe Network Display Page (Create Profile View Wizard)

Use this page to specify which pipe networks or parts to include in the profile view.

**NOTE** This page is not available if there are no pipe networks present in the drawing.

### Select Stacked View To Specify Options For

**NOTE** This control is not available if the Show Offset Profiles By Stacking Views check box on the General page (page 2146) is cleared.

Displays a list of the stacked profile views. The number of profile views in this list depends on the Number Of Stacked Views specified on the Stacked Profile page (page 2147). Select a profile view from the list, and then use the Select Pipe Networks To Draw In Profile View table to specify the pipe network display settings for that stacked profile view.

### Select Parts From Screen

Select pipe networks or parts by clicking in the drawing window. Click then follow the prompts at the command line.

### Select Pipe Networks to Draw in Profile View

Use this list to select pipe networks or parts from the drawing. By default, this list is populated with all pipe networks in the current drawing but none of the parts are selected.

Select All Networks
   Selects the Select check box for all pipe networks and parts listed in the table.

Deselect All Networks
   Clears the Select check box for all pipe networks and parts listed in the table.

Name
   Specifies the name of the pipe network and its parts.

Select
   Specifies whether a network or part is drawn in the profile view. Check box displays one of three states:

   Not Selected. If a part is not selected, it is not drawn. If a pipe network is not selected, none of its parts are drawn.

   Selected. If a part is selected, it is drawn. If a pipe network is selected, all of its parts are drawn.

   Partially selected. If a pipe network is partially selected, only its selected parts are drawn. Parts cannot be partially selected.

Show Only Parts Selected
   Specifies whether to display only the pipes and structures that are selected to draw in the profile view.

## Data Bands Page (Create Profile View Wizard)

Use this page to specify the properties of the data bands associated with the profile view.

Select Band Set
   Specifies the set of band styles for the profile view. Provides standard controls you use to review or change the set.

   **NOTE** This control is disabled when multiple profile views are created as part of the plan production process.

### List of Bands

Location
   Specifies either the top or bottom of the profile view. The table below this field shows current location of the bands in the current band style set.

**Band Type**

Specifies the data band type: Profile Data, Vertical Geometry, Horizontal Geometry, Superelevation, Sectional Data, or Pipe Data.

**Style**

Specifies the style for the band type.

**Profile 1/Profile 2**

Specifies which profile or profiles supply the data for the band. For example, if two separate profiles are listed in Profile 1 and Profile 2 columns, then each station displayed in the band shows the elevation information for both profiles.

**Alignment**

Specifies either the parent alignment of current profile view or the parent alignment of any superimposed profiles.

**Geometry Points**

Specifies the geometry points to label. Click [...] to open the Geometry Points to Label in Band dialog box (page 2162).

**Data Source**

Specifies the source of the data displayed in the band. The options displayed depend on the type of data band:

Pipe Data Band: Select a pipe network in the drawing file.

Sectional Data Band: Select a sample line group that is associated with the alignment in the profile view

This column is available only for pipe and sectional data bands.

**Material**

Specifies a material or list of materials for which data is displayed in a sectional data band. The options are generated using the properties of sample line groups that are associated with the alignment in the profile view.

# Hatch Options Page (Create Profile View Wizard)

Use this page to add hatch areas to show cut and fill regions in a profile view.

### Add Areas to Hatch

**Cut Area**

Click to add hatching to the profile view in areas where terrain must be cut to create the design profile.

**Fill Area**

Click to add hatching to the profile view in areas where terrain must be filled to create the design profile.

**Multiple Boundaries**

Click to add hatching to the profile view in areas where multiple boundaries are used to define a hatch region.

**From Criteria**

Click to add hatching to the profile view in areas where quantity takeoff criteria are used to define a hatch region.

**Hatch Table**

Hatch Area

Specifies the upper and lower boundaries for each hatch area. Use the buttons along the top of the table to manage these boundaries. Use the + and – buttons to expand and collapse the list of boundaries for each hatch area heading.

 Collapse All

Collapse the display of boundaries to show only the top-level headings.

 Expand All

Expand the display to show all boundaries.

Add Upper Boundary

Add an upper boundary to the selected hatch area.

Add Lower Boundary

Add a lower boundary to the selected hatch area.

 Delete

Delete the selected boundary.

Profile

Specifies the surface or layout profile that defines each hatch area boundary. Where only a single surface and single layout profile exist in the profile view, the profiles are automatically assigned to boundaries. Click the arrow beside the profile name to select a different profile.

Shape Style

Specifies the shape style applied to the hatch area. Click the name to select another style or modify the style properties. These styles are found on the Toolspace Settings tab ➤ General ➤ Multipurpose Styles ➤ Shape Styles.

## Multiple Plot Options Page (Create Profile View Wizard)

Use this page to specify the layout of the plotted profile view segments.

**NOTE** This page is disabled if you are creating a single profile view.

**Draw Order**

By Rows

Specifies to draw profile view segments in a row. After the number of views in the Maximum in a Row field has been reached, a new row is started.

By Columns

Specifies to draw profile view segments in a column. After the number of views in the Maximum in a Column field has been reached, a new column is started.

Maximum in a Row (or Column)

Specifies the maximum number of profile view segments to draw in a single row or column. After this value is reached, a new row or column is started.

Start Corner

Specifies the direction in which to draw the profile view segments:

**Lower Left**: Draw profile view segments in rows starting from the lower left. When the Maximum in a Row value is reached, a new row is started above.

**Upper Left**: Draw profile view segments in rows starting from the upper left. When the Maximum in a Row value is reached, a new row is started below.

**Lower Right**: Draw profile view segments in rows starting from the lower right. When the Maximum in a Row value is reached, a new row is started to the left.

**Upper Right**: Draw profile view segments in rows starting from the upper right. When the Maximum in a Row value is reached, a new row is started to the left.

### Gap Between Adjacent Profile Views

Row

Specifies amount of spacing (in plotted units) to leave between adjacent rows of profile view segments.

Column

Specifies amount of spacing (in plotted units) to leave between adjacent columns of profile view segments.

# Create Quick Profiles Dialog Box

Use this dialog box to create a temporary profile to view elevation information along a line, polyline, feature or lot line, survey figure, or surface.

### Surfaces to Sample

Select All Surfaces

Specifies whether to sample all surfaces in the drawing in the quick profile. This option is cleared when you clear any of the surfaces in the list.

Surface

The name of a surface in the drawing. This field is read only.

Select

Select the check box to sample the surface in the quick profile.

Profile Style

Specifies the style used to display the surface in the quick profile. To change the style, click in the Profile Style column to open the Pick Profile Style dialog box. Select a style in the list or use the standard controls to create a style, copy or edit the current style selection, or pick a style from the drawing.

Profile View Style

Specifies the style of the profile view. Select a style in the list or use the standard controls to create a style, copy or edit the current style selection, or pick a style from the drawing.

Draw 3D Entity Profile

Specifies whether the elevation data for the selected 3D object is displayed in the quick profile. Used only when the object you selected is a 3D object such as a 3D line or polyline.

3D Entity Profile Style

Specifies the style used to display the 3D object elevation data in the quick profile. Select a style in the list or use the standard controls to create a style, copy or edit the current style selection, or pick a style from the drawing.

**Related procedures:**

■  Creating a Quick Profile (page 1058)

# Curb Return Profile Properties Dialog Box

Use this dialog box to view or change the properties of a curb return profile, such as its name, style, and station data.

These properties apply to layout, surface, superimposed, and corridor profiles.

**See also:**

■  Profile Properties (page 1024)

## Information Tab (Curb Return Profile Properties Dialog Box)

Use this tab to view or change primary information about a curb return profile.

Name
Specifies the name of the profile. The application generates a default value, or you can enter a name.

Description
Specifies an optional description of the profile.

Object Style
Specifies the name of the style used by the profile. Use the buttons to preview or edit the style.

Show Tooltips
Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Profile Data Tab (Curb Return Profile Properties Dialog Box)

Use this tab to view structural data about a curb return profile.

Name
Specifies the name of the profile.

Description
Specifies an optional description for the profile.

Type
Specifies whether the profile is a surface profile ⌣ , layout profile ⌣ , superimposed profile, or corridor profile ⌣ .

Data Source
This column is not used for curb return profiles.

Offset
This column is set to 0.00 for curb return profiles.

Update Mode
This column is not used for curb return profiles.

Layer
Specifies the drawing layer on which the profile is placed.

Style
Specifies the style used by the profile.

Station Start/End
Specifies the first and last station numbers for the profile. The station numbers represent locations on the curb return alignment.

Elevation Minimum/Maximum
Specifies the highest and lowest elevation values that occur along the profile.

Alignment
Specifies the name of the curb return alignment for the profile.

## Design Criteria Tab (Curb Return Profile Properties Dialog Box)

Use this tab to apply or change the profile design criteria.

### Use Criteria-Based Design

Specifies whether to apply design criteria to the profile. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

### Use Design Criteria File

Use Design Criteria File
Specifies whether to associate a design criteria file with the profile. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.
For more information, see Profile Standards in the Design Criteria File (page 1045).

Design Criteria File Selector
Specifies the design criteria file to associate with the profile. The design criteria file that is applied to the parent alignment is displayed by default. Click ![button] to select another design criteria file.

Default Criteria Table
Displays the profile standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

### Use Design Check Set

Use Design Check Set
Specifies whether to associate a design check set with the profile. If this check box is cleared, then the Design Check Set list is not available.
For more information, see Profile Design Check Sets (page 1046).

Design Check Set List
Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

![icon] Design Check Set Selection
Edits or copies the current design check set, or creates a design check set.

## Offset Edge Profiles Tab (Curb Return Profile Properties Dialog Box)

Use this tab to view data about the other profiles attached to a curb return profile.

### Incoming Edge Information

Incoming Alignment
Specifies the alignment to which the start of the curb return alignment is attached.

Incoming Profile

    Specifies the profile to which the start of the curb return alignment is attached.

Incoming Profile Type

    Specifies the profile type as either dynamic cross slope, with a numeric slope value, or static.

**Outgoing Edge Information**

Outgoing Alignment

    Specifies the alignment to which the end of the curb return alignment is attached.

Outgoing Profile

    Specifies the profile to which the end of the curb return alignment is attached.

Outgoing Profile Type

    Specifies the profile type as either dynamic cross slope, with a numeric slope value, or static.

# Edit Feature Settings - Profile Dialog Box

Use this dialog box to view and change profile-related settings.

This topic documents settings in all profile-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the ⬜ drawing icon.

- Profile feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ⌄ profile icon.

- Profile command settings are identified by the ⬛ command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## ⌄ Default Styles

Use these settings to specify the default styles assigned to profiles and their labels.

Profile Label Set

    Specifies the default profile label style set. Click in the Value column, and click […] to select a style in the Profile Label Set dialog box.

Profile Style

    Specifies the default profile style. Click in the Value column, and click […] to select a style in the Profile Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## ⬛ Default Name Format

Use these settings to specify the default name formats for profiles, offset profiles, superimposed profiles, and 3D entity profiles. Click in the Value column, and click […] to make changes in the Name Template dialog box (page 1806).

## ⌄ Profile Creation

Use these settings to specify the default values for vertical curve sub-entities. For all linear settings such as length, height, or radius, enter a number in the Value column or click and select a distance in the drawing area.

Default Vertical Curve Type
    Specifies the default vertical curve type for a new profile:

- Circular Curve

- Symmetric Parabola

- Asymmetric Parabola

Parabolic Sag Curve Length And K-Value
    Specifies whether curve length or K value is the primary constraint on a parabolic sag curve.

Parabolic Crest Curve Length And K-Value
    Specifies whether curve length or K value is the primary constraint on a parabolic crest curve.

Parabolic Crest Curve Length
    Specifies the default length for a parabolic crest curve.

Parabolic Crest Curve K-Value
    Specifies the default K-value for a parabolic crest curve.

Parabolic Sag Curve Length
    Specifies the default length for a parabolic sag curve.

Parabolic Sag Curve K-Value
    Specifies the default K-value for a parabolic sag curve.

Circular Crest Curve Length And Radius
    Specifies whether curve length or radius is the primary constraint on a circular crest curve.

Circular Sag Curve Length And Radius
    Specifies whether curve length or radius is the primary constraint on a circular sag curve:

Circular Crest Curve Length
    Specifies the default length for a circular crest curve.

Circular Crest Curve Radius
    Specifies the default radius for a circular crest curve.

Passing Eye Height
    Specifies the default passing eye height.

Circular Sag Curve Length
    Specifies the default length for a circular sag curve.

Passing Object Height
    Specifies the default passing object height.

Circular Sag Curve Radius
    Specifies the default radius for a circular sag curve.

Stopping Eye Height
    Specifies the default stopping eye height.

Asymmetrical Crest Curve Length 1

Specifies the default length for the first part of the asymmetrical crest curve.

Stopping Object Height

Specifies the default stopping object height.

Asymmetrical Crest Curve Length 2

Specifies the default length for the second part of the asymmetrical crest curve.

Headlight Angle

Specifies the default headlight angle.

Asymmetrical Sag Curve Length 1

Specifies the default length for the first part of the asymmetrical sag curve.

Headlight Height

Specifies the default headlight height.

Asymmetrical Sag Curve Length 2

Specifies the default length for the second part of the asymmetrical sag curve.

## Criteria-Based Design Options

Use these settings to specify the default design criteria.

For more information, see Criteria-Based Profile Design (page 1044).

**NOTE** The design criteria values of the parent alignment take precedence over the default design criteria values of the profile. If the parent alignment does not use design criteria values, then the criteria-based design options in the profile are used.

Use Design Criteria File Option

Specifies whether to associate a design criteria file with the profile.
The first XML file found in the *<install directory>\Data\Corridor Design Standards\<units>* is used by default. You can select another design criteria file by using either the Create Profile - Draw New dialog box (page 2142) during profile creation, or the Profile Properties dialog box (page 2175) after the profile is created.

Use Design Checks Option

Specifies whether to associate a design check set with the profile.

Default Design Check Set

Specifies the default design check set. Click in the Value column, and click [⋯] to select a set in the Default Design Check Set dialog box.
You can select another design check set by using either the Create Profile - Draw New dialog box (page 2142) during profile creation, or the Profile Properties dialog box (page 2175) after the profile is created.

**NOTE** This setting is not available if the Use Design Checks Option is set to False.

## Curve Tessellation Option

**NOTE** This property group is displayed when accessing the settings from the CreateProfileLayout command.

Use these settings to specify the default behavior when curve entities are used as an input option for best fit arc, line, and parabola creation.

Tessellate Curve

Specifies whether to tesselate curve segments if they are included as an input option.

Mid-ordinate Tolerance

Specifies the mid-ordinate tolerance based on a distance across the tessellated curve segments. Enter a length in the Value column or click and select two points along a curve.

## Regression Graph Option

**NOTE** This property group is displayed when accessing the settings from the CreateProfileLayout command.

Use this setting to specify the default behavior for regression analysis graphs in best fit arc, line, and parabola creation.

Spline Fit For Regression Analysis Graph

Specifies whether the regression graph displays splines on the green line that represents the best fit entity.

## Quick Profile Options

**NOTE** This property group is displayed when accessing the settings from the CreateQuickProfile command.

Use these settings to specify the default settings for quick profile creation.

Select All Surfaces

Specifies whether to generate profiles of all surfaces in the drawing file in the quick profile.

Create Profile of 3D Entities

Specifies whether to include elevation data for the selected 3D entity in the quick profile.

## Superimpose Profile

**NOTE** This property group is displayed when accessing the settings from the SuperimposeProfile command.

Use these settings to specify the default values for the mid-ordinate distances that the system uses when tessellating (approximating) the shape of curves in superimposed profiles.

Horizontal Mid-ordinate Distance

Specifies the horizontal mid-ordinate distance for tessellated curve segments. Enter a distance in the Value column or click and select a distance in the drawing.

Vertical Mid-ordinate Distance

Specifies the vertical mid-ordinate distance for tessellated curve segments. Enter a length in the Value column or click and select a distance in the drawing.

These settings are used in the Superimpose Profile Options dialog box (page 2193).

**Related procedures:**

■ Profile Settings (page 1025)

■ Creating Profiles (page 1046)


# Edit Feature Settings - Profile View Dialog Box

Use this dialog box to view and change profile view-related settings.

This topic documents settings in all profile view-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the drawing icon.

- Profile view feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ![profile view icon] profile view icon.

- Profile view command settings are identified by the ![command icon] command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## ![icon] Default Styles

Use these settings to specify default styles assigned to profile views and their labels.

Marker Style
> Specifies the default style for markers associated with manually placed profile view labels for station and elevation or depth. Click in the Value column, and click ![...] to select a style in the Marker Style dialog box.

Profile View Style
> Specifies the default profile view style. Click in the Value column, and click ![...] to select a style in the Profile View Style dialog box.

Profile Label Set
> Specifies the default style for profile label sets. Click in the Value column, and click ![...] to select a style in the Profile Label Set dialog box.

First Split View Style
> Specifies the default style for the first split profile segment. Click in the Value column, and click ![...] to select a style in the First SplitView Style dialog box.

Intermediate Split View Style
> Specifies the default style for the intermediate split profile view segments. Click in the Value column, and click ![...] to select a style in the Intermediate SplitView Style dialog box.

Last Split View Style
> Specifies the default style for the last split profile view segment. Click in the Value column, and click ![...] to select a style in the Last SplitView Style dialog box.

Top Stack View Style
> Specifies the default profile view style of the top-most profile view in the stack. Click in the Value column, and click ![...] to select a style in the Top StackView Style dialog box.

Middle Stack View Style
> Specifies the default profile view style of all profile views between the top-most and bottom-most profile views. Click in the Value column, and click ![...] to select a style in the Middle StackView Style dialog box.

Bottom Stack View Style
> Specifies the default profile view style of the bottom-most profile view in the stack. Click in the Value column, and click ![...] to select a style in the Bottom StackView Style dialog box.

Profile Label Set

Specifies the default profile label set. Click in the Value column, and click ![icon] to select a set in the Profile Label Set dialog box.

Profile Station and Elevation Label Style

Specifies the default style for profile station and elevation labels. Click in the Value column, and click ![icon] to select a style in the Profile Station and Elevation Label Style dialog box.

Profile Depth Label Style

Specifies the default style for profile depth labels. Click in the Value column, and click ![icon] to select a style in the Profile Depth Label Style dialog box.

Projection Label Style

Specifies the default style for labels on projected objects. Click in the Value column, and click ![icon] to select a style in the Profile Projection Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use this setting to specify the default name format for profile views. Click in the Value column, and click ![icon] to make changes in the Name Template dialog box (page 1806).

## Profile View Creation

Use these settings to specify default parameters for the creation of profile views.

Specify Profile Station Range
   Specifies whether to allow the user to set range of stations to be displayed in the profile view.

Specify Profile Elevation Range
   Specifies whether to allow the user to set a minimum and maximum height to be displayed in the profile view.

Pipe Network Parts
   Specifies whether to Display Only Selected or to Display All pipe network parts in the profile view.

## Split Profile View Options

Use these settings to specify default parameters for splitting profile views.

**NOTE** If the Split Profile Views option is set to No, then the remaining Split Profile View Options are not available.

Split Profile Views
   Specifies whether to split a profile view that exceeds the maximum profile view height setting.

Split Station Option
   Specifies the default split station rounding setting.

Split Datum Option
   Specifies the default datum elevation rounding setting.

## Stacked Profile View Options

Use these settings to specify default parameters for stackng profile views.

**NOTE** If the Stacked Views option is set to No, then the remaining Stacked Profile View Options are not available.

Stacked Views

Specifies whether to create stacked profile views by default.

Number of Stacked Views

Specifies the default total number of profile views to create. This unitless value can be any integer from 2 to 9.

Gap Between Views

Specifies the default amount of spacing (in plotted units) to leave between stacked profile views. Enter a distance in the Value column or click and select a distance in the drawing. A negative value in this field causes the stacked profile views to overlap.

 **Multiple Profile View Creation**

NOTE  This property group is displayed when accessing the settings from the  CreateMultipleProfileView command.

Use these settings to specify defaults for creating multiple profile views.

Default Profile VIew Length

Specifies the default length for profile views. Enter a length in the Value column or click and select a length in the drawing area.

Default Profile VIew Height

Specifies the default height for profile views. Enter a height in the Value column or click  and select a height in the drawing area.

Plot Pattern

Specifies whether profile views are by default plotted by rows or columns.

Profile Views In Row Or Column

Specifies the default number of profile views in a row or column.

Start Corner

Specifies the default corner in which the first of the multiple profile views will be plotted. (Lower Left, Lower Right, Upper Left, Upper Right)

Column Spacing

Specifies the default gap between individual profile views in a column. Enter a distance in the Value column or click  and select a distance in the drawing area.

Row Spacing

Specifies the default gap between individual profile views in a row. Enter a distance in the Value column or click  and select a distance in the drawing area.

**Related procedures:**

■  Profile View Settings (page 1026)

# Geometry Points to Label in Band Dialog Box

Use this dialog box to specify the horizontal and vertical geometry points that you want to label in the profile view data band.

**See also:**

■  Creating Profile Views (page 1108)

## Alignment Points

Use this tab to specify which alignment geometry points you want to label in the data band.

### Select Geometry Points To Label

 **Select All**

Selects all check boxes in the Label column.

 **Deselect All**

Clears all check boxes in the Label column.

Geometry Point Column

Identifies the available geometric details.

> **NOTE** The Abbreviations tab (page 1855) in the Drawing Settings dialog box also lists these geometry points.

Label Column

Specifies whether to label a geometry point.

## Profile Points

Use this tab to specify which profile geometry points you want to label in the data band.

### Select Geometry Points To Label

 **Select All**

Selects all check boxes in the Label column.

 **Deselect All**

Clears all check boxes in the Label column.

Geometry Point Column

Identifies the available geometric details.

> **NOTE** The Abbreviations tab (page 1855) in the Drawing Settings dialog box also lists these geometry points.

Label Column

Specifies whether to label a geometry point.

# Import Quantity Takeoff Criteria Dialog Box

Use this dialog box to specify profile view hatch areas based on quantity takeoff criteria.

Pick quantity takeoff criteria to import

Select a criteria from the list. Criteria defined for the drawing are specified on the Toolspace Settings tab ➤ Quantity Takeoff ➤ Quantity Takeoff Criteria. Standard buttons are provided to create or edit a style. For more information, see Select Style dialog box (page 1805).

**Hatch Boundary Table**

Area Boundary Name
Specifies the boundaries defined for the quantity takeoff criteria.

Profile
Specifies the profile in the profile view that is assigned to the boundary.

**Related procedures:**

■ Adding Hatch Areas to a Profile View (page 1116)


# Insert PVIs Dialog Box

Use this dialog box to record data about points of vertical intersection (PVIs) that you want to add to a layout profile.

After you select the vertical curve type (or None), the table grid shows the data columns that apply to that type.

The dialog box is designed to support rapid entry of PVI data from the numeric keypad. In the table grid, after you enter the first station number, press Enter to move to the next cell, and continue in this manner.

**Related procedures:**

■ Editing PVIs (page 1093)


# Profile Data Band Style Dialog Box

Use this dialog box to create band styles that annotate profile and alignment data on the profile views.

Create separate band styles for profile data (station and elevation), vertical geometry, horizontal geometry, superelevation, pipe data, and sectional data. The procedures are similar for each type.

**See also:**

■ Profile View Band Styles (page 1038)
■ Editing Profile Views (page 1113)


# Information Tab (Profile Data Band Style Dialog Box)

Use this tab to change the name and description of a band style or review details, such as when the style was most recently modified.

Name
Specifies the name of the band set.

Description
Specifies an optional description of the band set.

Created By
Displays the Windows login name of the person who created the band set.

Date Created
Displays the date and time the band set was created.

Last Modified By
    Displays the Windows login name of the person who last modified the band set.

Date Modified
    Displays the date and time the band set was last modified.

# Band Details Tab (Profile Data Band Style Dialog Box)

Use this tab to design the format and contents of the band style.

## Title Text

Click the Compose Label button to change attributes of the band title, using the Label Style Composer dialog box (page 1942).

## Layout

Band Height
    Specifies the vertical dimension of the band in plotted units, which can affect readability of the contents.

Text Box Width
    Specifies the width (in plotted units) of the box that contains the band title.

Offset From Band
    Specifies the distance between the title box and the left side of the data band.

Text Box Position
    Specifies the location of the title text box with respect to the band: Left of Band or Right of Band.

## Labels and Ticks

At
    Specifies the location type to be labeled, which varies according to the band style:
    **Profile Data Bands Style:** major station, minor station, horizontal geometry point (HGP), vertical geometry point (VGP), station equation, or incremental distance.

    **Vertical Geometry Bands Style:** uphill tangent, downhill tangent, crest curve, or sag curve.

    **Horizontal Geometry Bands Style:** tangent, curve, or spiral.

    **Superelevation Data Bands Style:** normal crown, level crown, reverse crown, full super, shoulder critical points, or slope transition region.

    **Sectional Data Bands Style:** sample line station, incremental section data.

    **Pipe Data Bands Style:** structure, pipe.

    Several label styles can appear in the same data band, but each one must be composed separately. Each appears in the preview as it is created.

Full Band Height Ticks
    Select if you want a line across the band for each tick.

Small Ticks At
    Select if you want smaller ticks. Select the tick location (Top, Middle, or Bottom of the band), and specify the tick size in drawing units.

Compose Label
    Click to review or change the label text, using the Label Style Composer dialog box (page 1942).

Schematic Line Option
    For horizontal geometry data bands, specify the information for schematic lines to draw: Geometry, Radius, or Curvature.

## Display Tab (Profile Data Band Style Dialog Box)

Use this tab to manage the display of profile view band style components.

Each type of band style has a slightly different set of display components, but they are all managed in the same way. For more information about the Component display section of this tab, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Profile Data Band Style Dialog Box)

Use this tab to review and adjust settings for the profile view band style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Profile Entities Vista

Use this vista to view and edit numeric data about curves and tangents at a PVI or for a specific sub-entity in a profile.

If ⟋ PVI Based is selected in the Profile Layout Tools (page 2170), the vista displays PVI data. If ⟋ Entity Based is selected, the vista displays entity data.

Use the Profile Entities vista to navigate through all entities that make up the profile. You can select an entity in the Profile Entities vista and then edit it in the Profile Layout Parameters dialog box (page 2168). Attributes in the Profile Entities vista and Profile Layout Parameters window are automatically updated as the profile is edited.

If design criteria has been applied to the profile, a ⚠ warning symbol is displayed next to sub-entities that violate the specified design criteria. For more information, see Viewing and Correcting Profile Design Criteria Violations (page 1103).

The display can be filtered to show a selected range of entities along a profile. See Selecting Profile Sub-entities (page 1092) for more information.

The Profile Entities vista is like a spreadsheet. Each row represents a profile entity and each column represents an attribute for that entity. You can resize, move, and copy columns.

The first column (No.) is pinned and always displayed. It does not move when you use the scroll bar.

Two column configurations are available: default and criteria-based design. To switch between the configurations, or to create your own configuration, right-click a column heading, and click Customize Columns.

This window remains open when you grip edit the profile or enter another command. The window closes when you either delete the currently selected profile, click the X button, or click 🖳 Profile Grid View on the Profile Layout Tools.

**Related procedures:**

■ Vertical Curve Design (page 1068)

■ Criteria-Based Profile Design (page 1044)

■ Selecting Profile Sub-entities (page 1092)

■ Editing Profile Curve and Tangent Parameter Values (page 1098)

# Profile Label Set Dialog Box

Use this dialog box to manage the labels for a particular profile in a profile view. A set of labels created here can be saved and used for other profiles.

Type
Specifies the type of label to be added to the profile in the profile view: either major stations, minor stations, horizontal geometry points, lines, grade breaks, sag curves, or crest curves.

Profile <Label Type> Label Style
Specifies the style for the labels. Standard buttons are provided to create or edit a label style.

Add >>
Adds the specified label type to the set in the table below. These labels are displayed along the profile line in the profile view.

✖
Deletes the selected label type from the table, and also removes it from the profile.

## Label Table

Type
Specifies the type of profile label.

Style
Specifies the label style.

Increment
Specifies the distance between labels of this type; used only for station labels.

Check
Specifies whether the labels are applied from the beginning of the profile or start at a specified station.
**Selected**: The label type is applied from the beginning of the profile.
**Cleared**: The label type is applied to the part of the profile that begins at the station recorded in the Start Station column.

Start Station
Specifies the station where the label type begins.

Check
Specifies whether the labels are applied to the end of the profile or finish at a specified station.
**Selected**: The label type is applied as far as the end of the profile.
**Cleared**: The label type is applied to a part of the profile that ends at the station recorded in the End Station column.

End Station
Specifies the station where the label type ends.

Dim Anchor Opt
This option is for crest and sag curve labels only. Specifies the location of the dimension line that marks the length of the vertical curve. Uses the distance specified in Dim Anchor Val. Select one of the following:
**Distance Above**: Places the line a specific distance above the curve.
**Distance Below**: Places the line a specific distance below the curve.
**Fixed Elevation**: Places the line at a fixed elevation on the profile view.
**Graph View Top**: Places the line anchor at the top of the profile view grid. This option creates a vertical graph line above the curve label. To display the full extent of the line, set Dim Anchor Val to 0.

**Graph View Bottom**: Anchors the line to the bottom of the profile view grid. This option creates a vertical graph line below the curve label. To display the full extent of the line, set Dim Anchor Val to 0.

Dim Anchor Val

Specifies the vertical distance or elevation used with the setting in Dim Anchor Opt.

Weeding

For grade break labels only. Remove overlapping labels making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance. From a given grade break, if another break occurs within the specified distance, it is not labeled.

Geometry Points To Label

For Horizontal Geometry Points label types, specifies the alignment geometry points to label. Click [...] to open the Alignment Geometry Points dialog box (page 1739).

Stagger Labels

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left.

---

**NOTE** The staggering options are available for the following labels: Profile (Grade Breaks, Major/Minor Station, Horizontal Geometry), and Section (Major/Minor Offsets and Grade Break).

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Stagger Line 1 Height

Specifies the height of the first label stagger line.

Stagger Line 2 Height

Specifies the height of the second label stagger line.

**Related procedures:**

■ Profile Labels and Label Styles (page 1033)

■ Staggering Profile and Section Labels (page 1527)

# Profile Layout Parameters Dialog Box

Use this dialog box to analyze and modify profile data in a layout profile.

When the Profile Entities Vista (page 2166) is open, click any table row of PVI or sub-entity data. The data populates the Profile Layout Parameters dialog box in a vertical layout that you may find more convenient for analysis.

If ⬧ PVI Based is selected in the Profile Layout Tools (page 2170), the Profile Layout Parameters dialog box displays PVI data. If ⬧ Entity Based is selected, the Profile Layout Parameters dialog box displays entity data.

You can edit available parameters. You cannot edit parameters that are unavailable (shaded).

If the profile uses design criteria, then the Profile Layout Parameters dialog box contains up to three panels that can be collapsed by clicking ⬧ . If design criteria has not been associated with the profile, then only the Layout Parameters panel is available. If a sub-entity violates the specified design criteria, a ⚠ warning symbol is displayed next to sub-entities that violate the specified design criteria. For more information, see Viewing and Correcting Profile Design Criteria Violations (page 1103).

You cannot dock the Profile Layout Parameters dialog box, but you can pin or resize it.

This window remains open when you grip edit the profile or enter another command. The window closes when you either delete the currently selected profile, click the X button in the upper right-hand corner of the window, or click  on the Profile Layout Tools (page 2170) toolbar.

**Design Speed**

Specifies the design speed of the current sub-entity. The design speed is set in the properties of the parent alignment.

**NOTE** If multiple speeds are assigned to a sub-entity, then the highest speed is used to look up constraint values from the design criteria file and to validate the design checks. This ensures the safest design for that sub-entity.

**Design Criteria Panel**

The following parameters are displayed only if design criteria has been associated with the profile.

Use the Design Criteria panel to apply design criteria that is different from the default values that have been set for the profile. To change a criterion value, click the Value cell in the appropriate row.

A  warning symbol in the Property column indicates that the sub-entity design violates the criteria set in the design criteria file. You can find the specific parameter that has been violated by examining the Layout Parameters panel.

Minimum K For Stopping Sight Distance

Specifies the minimum K value for stopping sight distance at the specified design speed.

Minimum K For Passing Sight Distance

Specifies the minimum K value for passing sight distance at the specified design speed.

Minimum K For Headlight Sight Distance

Specifies the minimum K value for headlight sight distance at the specified design speed.

**Layout Parameters**

Use this panel to examine and change the general layout parameters of the selected sub-entity.

Click  Show More to expand the number of parameters that appears in the window. The default, collapsed view displays editable and other important parameters. Use the horizontal and vertical scroll bars when the list is longer or wider than the window.

When a design criteria file has been associated with a profile, the Constraints column displays the value to which a given parameter is limited. The selected design criteria file defines the constrained values. When a parameter value violates the range displayed in the Constraints column, a  warning symbol appears next to the parameter name.

**NOTE** The constraint definition for each profile entity determines the parameters that are displayed in the Layout Parameters panel.

**Design Checks**

The Design Checks panel displays the name and contents of the design check set that is associated with the profile.

When the sub-entity design violates a design check, a  warning symbol appears next to the affected design check.

**NOTE** The Design Checks panel is available only for profiles that use a design check set.

**Related procedures:**

■ Vertical Curve Design (page 1068)

■ Editing Profile Curve and Tangent Parameter Values (page 1098)

■ Criteria-Based Profile Design (page 1044)

# Profile Layout Tools

Use the tools in this toolbar to draw and edit profiles and to analyze vertical tangents and curves.

These tools are most useful for editing profiles by adding tangents and curves, or moving PVIs.

For dynamic surface profiles, most tools in this toolbar are not available. Use the Profile Entities Vista (page 2166) to display profile data. Similarly, many of the tools are unavailable for use with locked PVIs, including the endpoints of curb return profiles.

The name of the currently selected profile is displayed at the top of the toolbar. When you click an icon, the current command is displayed at the bottom of the toolbar. The toolbar remains open if you grip edit the profile or enter another command. The toolbar closes if you either delete the currently selected profile or click the X button in the upper right-hand corner of the toolbar.

### Layout Profile Creation

These three buttons control how a layout profile is drawn on a profile view.

Draw Tangents

Adds a series of fixed tangents between specified points.

Draw Tangents With Curves

Adds a series of fixed tangents between specified points, with free curves automatically added at the points of vertical intersection.

Curve Settings

Specifies the curve parameters to use with the Draw Tangents with Curves command.

To change the default curve settings, go to the Profile Creation section of the Edit Feature Settings - Profile (page 2156) dialog box.

### PVI Edit Operations

Use these three buttons to graphically modify the PVIs in a layout profile.

Insert PVI

Breaks a tangent into two adjacent tangents by creating a point of vertical intersection (PVI) at a specified location on the profile view.

Click above or below an existing tangent to specify the new PVI location.

Delete PVI

Creates a single tangent from two adjacent tangents by removing a point of vertical intersection (PVI) from a profile.

If two tangents meet at the PVI, they are replaced by a single tangent between two adjacent PVIs.

Move PVI

Moves a profile point of vertical intersection (PVI) to a new location on a profile view.

The tangents connected to the PVI remain attached. You can also use editing grips to move a PVI.

## Tangent Creation Operations

Use these three buttons to add tangents to a layout profile.

Fixed Tangent (Two Points)

Adds a fixed tangent between two specified points.

Fixed Tangent - Best Fit

Adds the most probable fixed tangent through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

Floating Tangent (Through Point)

Adds a floating tangent from an existing entity to a specified pass-through point.

Float Tangent - Best Fit

Adds the most probable floating tangent from a point on an existing entity through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entity, regardless of how the entity is edited.

Free Tangent (Between Two Parabola)

Adds a free tangent between two parabolic curves.

## Curve Creation Operations

Use these buttons to add a curve to a profile. You can create curves by selecting entities to attach to, specifying parameters, or selecting a PVI where you want to add a curve. Parameters can be specified numerically or graphically.

For example, to numerically specify a pass-through point, enter the point coordinates. To graphically specify the point, select a point in the drawing window.

To graphically specify a curve length, K value, or radius, use the cursor to pick two points on the screen the correct distance apart. For parabolic and circular curves, you see a preview of the curve as you move the cursor. For asymmetrical curves, you do not see the preview until you begin to specify the second length.

Fixed Vertical Curve (Three Points)

Adds a fixed curve through three specified points.

Fixed Vertical Curve (Two Points, Parameter)

Adds a fixed sag or crest curve, with a specified K value or minimum radius, through two specified points.

Fixed Vertical Curve (Entity End, Through Point)

Adds a fixed curve from the end of an existing entity to a specified pass-through point.

Fixed Vertical Curve (Two Points, Grade at Start Point)

Adds a fixed curve that is defined by two specified pass-through points and a specified grade at the start point.

Fixed Vertical Curve (Two Points, Grade at End Point)

Adds a fixed curve that is defined by two specified pass-through points and a grade at the end point.

**Fixed Vertical Curve - Best Fit**

Adds the most probable fixed vertical curve through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen.

**Floating Vertical Curve (Through Point, Parameter)**

Adds a floating curve, which is defined by either a K value or minimum radius, between an existing entity and a specified pass-through point.

**Floating Vertical Curve (Through Point, Grade)**

Adds a floating curve, which is defined by a specified pass-though point and grade, to an existing entity.

**Floating Vertical Curve - Best Fit**

Adds the most probable floating vertical curve from an existing entity through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entity, regardless of how the entity is edited.

**Free Vertical Curve (Parameter)**

Adds a free curve, which is defined by a specified curve length, radius, or K value, between two entities.

**Free Vertical Parabola (PVI Based)**

Adds a free vertical parabola, which is defined by a specified curve length, pass-through point, or K value, to an existing point of vertical intersection (PVI).

**Free Asymmetrical Parabola (PVI Based)**

Adds a free asymmetrical parabola, which is defined by specified before and after PVI curve lengths, to an existing point of vertical intersection (PVI).

**NOTE** If the first length you specify is too large for the asymmetric curve, you cannot enter the length for the second curve. The length of the first curve must be less than the station distance between the PVI you select and the station value of the previous PVI (or end station of the previous curve).

**Free Circular Curve (PVI Based)**

Adds a free circular curve, which is defined by a specified radius or pass-through point, to an existing point of vertical intersection (PVI).

**Free Vertical Curve - Best Fit**

Adds the most probable free vertical curve between two existing entities, and through a series of AutoCAD Civil points, AutoCAD points, existing entities, or clicks on screen. Tangency is maintained to the attached entities, regardless of how the entities are edited.

## Convert AutoCAD Line and Spline

This button enables you to convert an AutoCAD line or spline to a profile sub-entity.

**Convert AutoCAD Line and Spline**

Creates a fixed tangent or three-point vertical curve entity from an AutoCAD line or spline.

## PVI Operations

These buttons control various operations with PVIs when the Profile Grid View dialog box is active. If the profile is dynamic, only the Copy Profile button is available.

**Insert PVIs – Tabular**

Creates points of vertical intersection (PVIs) at multiple locations simultaneously.

**Raise/Lower PVIs**

Changes the vertical position of a series of points of vertical intersection (PVIs) simultaneously.

**Copy Profile**

Copies all or a part of a layout profile or static surface profile.

## Selection Operations

**PVI Based**

Edits the profile PVI parameter values in tabular format.

**Entity Based**

Edits the profile entity parameter values in tabular format.

**Select PVI**

Displays a selected PVI's parameter values in the Profile Layout Parameters dialog box. If you do not see this button, click ⟋ to change to a PVI-based view.

**Select Entity**

Displays a selected profile subentity's parameter values in the Profile Layout Parameters dialog box. If you do not see this button, click ⟋ to change to an entity-based view.

## Delete Entities

This button enables you to delete a tangent or curve.

**Delete Entity**

Deletes a specified tangent or curve. If you do not see this button, click ⟋ to change to an entity-based view.

## Data Analysis

**Edit Best Fit Data For All Entities**

Toggles the display of a horizontal table of regression data for all entities in the profile that were created by best fit.

**Profile Layout Parameters**

Toggles the display of a vertical table of numeric data about a single, selected profile subentity or PVI.

**Profile Grid View**

Toggles the display of a horizontal table of numeric data about multiple, selected profile sub-entities or PVIs.

**Undo**

Reverses the last AutoCAD Civil or AutoCAD command.

 Redo

Reverses the last AutoCAD Civil or AutoCAD undo operation. Redo is limited to one operation.

**Related procedures:**

■ Understanding the Profile Object (page 1019)

■ Creating Profiles (page 1046)

■ Editing Layout Profiles (page 1091)

# Profile Properties Dialog Box

Use this dialog box to view or change the properties of a profile, such as its name, style, and station data.

These properties apply to layout, surface, superimposed, and corridor profiles.

**See also:**

■ Profile Properties (page 1024)

## Information Tab (Profile Properties Dialog Box)

Use this tab to view or change primary information about a profile. These properties apply to layout, surface, superimposed, and corridor profiles.

Name

Specifies the name of the profile. The application generates a default value, or you can enter a name.

Description

Specifies an optional description of the profile.

Object Style

Specifies the name of the style used by the profile. Use the buttons to preview or edit the style.

Show Tooltips

Specifies whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Profile Data Tab (Profile Properties Dialog Box)

Use this tab to view structural data about a profile. These properties apply to layout, surface, superimposed, and corridor profiles.

Name

Specifies the name of the profile.

Description

Specifies an optional description for the profile.

Type

Specifies whether the profile is a surface profile , layout profile , superimposed profile, or corridor profile .

Data Source

For surface profile, displays the name of the surface represented in the profile. For superimposed profiles, displays the name of the superimposed profile. For corridor profiles, specifies the name of the source corridor. Not used for layout profiles.

Offset

Specifies the offset distance from the centerline of the parent horizontal alignment: either 0 for a centerline profile, a positive number for a right offset, a negative number for a left offset.

Update Mode

For surface profiles only. Specifies whether the profile updates automatically to reflect changes in surface elevation.

- **Dynamic**: The profile automatically updates to reflect changes in the surface elevation or the geometry of the parent horizontal alignment.

- **Static**: The profile shows surface elevations at the time of its creation. It does not update to reflect changes that occur later.

Layer

Specifies the drawing layer on which the profile is placed.

Style

Specifies the style used by the profile.

Station Start/End

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

Elevation Minimum/Maximum

Specifies the highest and lowest elevation values that occur along the profile.

Alignment

For superimposed profiles only. Specifies the horizontal alignment for the profile.

## Parent Horizontal Alignment

Name

Specifies the name of the horizontal alignment associated with this profile.

Stationing

Specifies the start and end station values for the horizontal alignment.


# Design Criteria Tab (Profile Properties Dialog Box)

Use this tab to apply or change the profile design criteria.

## Use Criteria-Based Design

Specifies whether to apply design criteria to the profile. If this check box is cleared, then the Use Design Criteria File and Use Design Check Set options are not available.

## Use Design Criteria File

Use Design Criteria File

Specifies whether to associate a design criteria file with the profile. If this check box is cleared, then the design criteria file selector and Default Criteria table are not available.

For more information, see Profile Standards in the Design Criteria File (page 1045).

Design Criteria File Selector

Specifies the design criteria file to associate with the profile. The design criteria file that is applied to the parent alignment is displayed by default. Click [···] to select another design criteria file.

Default Criteria Table

Displays the profile standards formulas that are defined in the selected design criteria file. Click the Value column to change a criteria table.

**Use Design Check Set**

Use Design Check Set

Specifies whether to associate a design check set with the profile. If this check box is cleared, then the Design Check Set list is not available.
For more information, see Profile Design Check Sets (page 1046).

Design Check Set List

Displays the default design check set. You can either accept the default design check set, or use the list to select a different one from the drawing.

Design Check Set Selection

Edits or copies the current design check set, or creates a design check set.

# Profile Style Dialog Box

Use this dialog box to define profile styles that control the display and behavior of profiles.

**See also:**

■ Profile Styles (page 1027)

# Information Tab (Profile Style Dialog Box)

Use this tab to change the style name and description and to review details such as when the style was most recently modified.

For more information, see Information Tab (Style Dialog Box) (page 1801).

# Design Tab (Profile Style Dialog Box)

Use this tab to set the representation of 3D curves.

**3D Chain Visualization**

Curve Tessellation Distance

Specifies the distance between elevation samples in the three-dimensional display of alignment data. Enter a small number to see a smooth 3D curve, or a large number to see a rougher approximation.

# Markers Tab (Profile Style Dialog Box)

Use this tab to specify the appearance of markers for locations on the profile.

**Profile Points**

Name

Displays the name of the point on the profile.

Marker Style

Specifies the style for the point marker on the profile. Click to open the Select Marker Style dialog box where you can select a style or <None>.

## Arrowhead

Type

Specifies the arrowhead style for the profile direction arrow. Select a style from the list or select User Arrow to select an AutoCAD block.

Fit Options

Specifies how the arrow is drawn when the profile segment is shorter than the arrowhead. The arrow may be reduced to a size 80 per cent of the segment length, not displayed, or displayed at its normal size. Select Shrink, Omit, or Always Draw.

Size Options

Specifies the method used to determine the size of the arrowhead:
**Use Drawing Scale**: Determines size of the arrowhead by multiplying the specified value by the drawing scale. Enter the scale factor.

**Use Fixed Scale**: Activates the Fixed Scale options.

**Use Size in Absolute Units**: Specifies that the arrowhead size is an absolute value based on the displayed units. Enter the value.

**Use Size Relative to Screen**: Specifies that the size of the arrowhead is a percentage of the drawing screen size. Enter the percentage.

Size

Specifies the size of the arrowhead in the units specified in Size Options.

Fixed Scale

Specifies independent fixed scale values when Size Options is set to Use Fixed Scale. Enter values for X, Y, and Z.

# Display Tab (Profile Style Dialog Box)

Use this tab to set the visibility and format of profile components, such as lines, curves, and arrows.

Profile components include:

**Line**: Solved lines with true tangent lengths that are part of the profile geometry.

**Circular Curve**: Solved, circular curves that are true curve lengths within the profile.

**Symmetrical Parabola**: Solved, symmetrical parabola that are true curve lengths within the profile.

**Asymmetrical Parabola**: Solved, asymmetrical parabola that are true curve lengths within the profile.

**Arrow**: Shows the direction of the profile entities, either solved or unsolved.

**Line Extensions**: Line segments that extend beyond the solved portion of a line to a pass-through point or another constraint.

**Parabolic Curve Extensions**: Parabolic curve segments that extend beyond the solved portion of a parabolic curve to a pass-through point or another constraint.

**Warning Symbol**: Markers that identify entities that violate the rules specified in the design criteria file. This component only applies to profiles that use design criteria.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Profile Style Dialog Box)

Use this tab to review and adjust settings for the profile style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Profile View Properties Dialog Box

Use this dialog box to view or change the properties of a profile view, including annotation, projected objects, and hatching.

**See also:**

■ Profile View Properties (page 1024)

## Information Tab (Profile View Properties Dialog Box)

Use this tab to view or change primary reference information about a profile view.

Name

Specifies the name of the profile view. Default value is PV - (n), where n is an integer that increments as profile views are created in the drawing.

Description

Specifies an optional description of the profile view.

Object Style

Specifies the style used by the profile view. Click the down arrow to select a style by name. Use the buttons to preview or edit the style.

**NOTE** If the profile view is split, this setting is ignored. Set split profile view styles on the Elevations tab (page 2179).

Show Tool tips

Controls whether tool tips are displayed for the object in the drawing (not over toolbar icons).

## Stations Tab (Profile View Properties Dialog Box)

Use this tab to view and change the horizontal range of the profile view.

### Alignment Name

Specifies the name of the alignment from which the profile view was created.

### Station Range

Automatic

Specifies the first and last stations of the parent horizontal alignment.

User Specified Range

Specify a station range if you want the profile view to show less than the full length of the horizontal alignment. Enter numeric station values or click  and pick points along the alignment in the drawing. If you specify the station range, the profile view length remains constant. If the alignment length is modified, the profile length does not adjust.

# Elevations Tab (Profile View Properties Dialog Box)

Use this tab to view and change the vertical range and split settings of the profile view.

See The Profile View Object (page 1021) for more information about split profile views.

### Elevation Range

Automatic Height

Specifies the elevation range for all profiles associated with the parent horizontal alignment, regardless of whether they are drawn in the profile view. The system refers to these values to set the default elevation range for the profile view.

User Specified Height

Specify an elevation range if you want the profile view to show other than the full height of the horizontal alignment.
If the current profile view style includes a padding (buffer) setting, it is applied to expand the elevation range. The padding is set on the Grid tab (page 2186) of the Profile View Style dialog box.
Selecting User-Specified Height enables the Split Profile View controls.

### Split Profile View

Selecting Split Profile View enables the Automatic and Manual controls.

Manual

Splits the profile view at user-specified locations and allows using a different style for each split segment.
**No:** Sequential number of split segments.

**Split Station:** Starting station value of each split location. The split station of segment number 1 is the starting station of the profile view and is not editable. Enter a specific value or click to select a location on the profile view.

**Adjusted Datum:** Vertical split location. Enter a specific value or click  to select a location on the profile view.

**Profile View Style:** Style selection of each profile view segment.

 Add a split location.

 Delete a split location.

Automatic

Splits the profile view to fit it within the user-specified height, rounding to specified station and datum locations.
**Station Rounding**: Specifies where to round the horizontal split location (Previous Major Grid, Previous Minor Grid, or Exact Elevation)

**Datum Rounding**: Specifies where to round the vertical split location (Previous Major Grid, Previous Minor Grid, or Exact Elevation)

**NOTE** Switching from manual to automatic clears all user-specified split settings.

# Profiles Tab (Profile View Properties Dialog Box)

Use this tab to view and change properties of the individual profiles in a profile view. Change column widths in the table to make it easier to see the contents of cells.

Name

Displays the name of the profile.

**Draw**

Specifies whether the profile is drawn in the profile view. After selecting or clearing check boxes here, click Apply to see the changes.

**Clip Grid**

Specifies which profile controls the size of the profile view grid.

**Split At**

Specifies which profile to favor when splitting a profile view containing multiple profiles.
For example, a profile view often contains both existing ground and finished ground profiles. Selecting Split At for the finished ground profile ensures that split occurs at the finished ground profile elevation.

**Description**

Specifies an optional description of the profile.

**Type**

Specifies whether the profile is a surface profile ⟋, layout profile ⋎ , superimposed profile ⋈, or corridor profile ⋎ .

**Data Source**

For surface profile, displays the name of the surface represented in the profile. For superimposed profiles, displays the name of the superimposed profile. For corridor profiles, specifies the name of the source corridor. Not used for layout profiles.

**Offset**

Displays the profile offset distance from the centerline of the parent horizontal alignment: either 0 for a centerline profile, a positive number for a right offset, or a negative number for a left offset.

**Update Mode**

For surface profiles only. Specifies whether the profile is dynamic or static.
**Dynamic**: The profile automatically updates to reflect changes in the surface elevation or the geometry of the parent horizontal alignment.

**Static**: The profile shows surface elevations at the time of its creation. It does not update to reflect changes that occur later.

**Layer**

Specifies the drawing layer on which the profile is placed.

**Style**

Specifies the style used by the profile. Click to open the Pick Profile Style dialog box where you can select a style.

**Override Style**

Specifies an override style for the display of the profile. This style is used only in the current profile view. Any other profile view that is plotting the profile would use the style as specified by the Style column value. Click the check box to open the Pick Profile Style dialog box where you can select a style.

**Labels**

Click to open the Profile Labels dialog box where you can edit the profile labels.

**Station Start/End**

Specifies the first and last station numbers for the profile. The station numbers represent distances along the parent horizontal alignment.

**Elevation Minimum/Maximum**

Specifies the highest and lowest elevation values that occur along the profile.

Alignment

For superimposed profiles only. Specifies the horizontal alignment for the profile.

# Bands Tab (Profile View Properties Dialog Box)

Use this tab to manage the data band sets for the profile view.

Band Type

Specifies which type of data band to add to the set: Profile Data, Vertical Geometry, Horizontal Geometry, Superelevation, Sectional Data, or Pipe Data.

Select Band Style

Specifies a style for the band. Standard buttons are provided to create or edit a style. For more information, see Select Style dialog box (page 1805).

Add >>

Adds the specified band type to the set for this profile view. Before clicking this button, ensure that the settings are correct for data band type, style, and location.

## List of Bands

Location

Specifies either the top or bottom of the profile view. The table below this field shows current location of the bands in the current band style set.

Band Type

Specifies the data band type: Profile Data, Vertical Geometry, Horizontal Geometry, Superelevation, Sectional Data, or Pipe Data.

Style

Specifies the style for the band type.

Description

Specifies optional descriptive notes about the band type

Gap

Specifies the distance between the current data band and the adjacent profile view grid or data band.

Geometry Points

For Profile Data bands, specifies the alignment and profile geometry points to label in the data band. Click [...] to open the Geometry Points to Label in Band dialog box (page 2162).

Label Start Station

Specifies whether the starting station of the band is labeled.

Label End Station

Specifies whether the ending station of the band is labeled.

Alignment

Specifies either the parent alignment of current profile view or the parent alignment of any superimposed profiles.

Profile1/Profile2

Specifies which profile or profiles supply the data for the band. For example, if two separate profiles are listed in Profile 1 and Profile 2 columns, then each station displayed in the band shows the elevation information for both profiles.

## Data Source

Specifies the source of the data displayed in the band. The options displayed depend on the type of data band:

Pipe Data Band: Select a pipe network in the drawing file.

Sectional Data Band: Select a sample line group that is associated with the alignment in the profile view

This column is available only for pipe and sectional data bands.

## Material

Specifies a material or list of materials for which data is displayed in a sectional data band. The options are generated using the properties of sample line groups that are associated with the alignment in the profile view.

## Weeding

Used only for labels at vertical geometry points on profile data bands. Vertical geometry points or grade breaks that are closer than the weeding factor are removed, making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance.

## Stagger Labels

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left. This option is only enabled for profile or section data band styles.

## Stagger Line Height

Specifies the height of the label stagger line.

## Band Position Controls

Changes the position of data bands in the set and deletes unwanted bands.

⇧ Move selected data bands up.

⇩ Move selected data bands down.

✖ Delete selected bands.

## Import Band Set

Opens the Band Set dialog box (page 2139). Select an existing band set to add to the band set for the current profile view.

## Save As Band Set

Opens the Band Set dialog box, where you can save the current band set for use with other profile views.

## Match major/minor increments to vertical grid intervals

Specifies if the profile data band major/minor interval distances for profile views matches the profile view style's major/minor grid spacing intervals. Select the checkbox to match major/minor increments to vertical grid intervals. If this checkbox is selected, the major/minor interval fields in the List of Bands table are unavailable.

**Related procedures:**

■ Adding Data Bands to a Profile View (page 1115)

# Hatch Tab (Profile View Properties Dialog Box)

Use this page to add hatch areas to show cut and fill regions in a profile view.

### Add Areas to Hatch

## Cut Area

Click to add hatching to the profile view in areas where terrain must be cut to create the design profile.

Fill Area

Click to add hatching to the profile view in areas where terrain must be filled to create the design profile.

Multiple Boundaries

Click to add hatching to the profile view in areas where multiple boundaries are used to define a hatch region.

From Criteria

Click to add hatching to the profile view in areas where quantity takeoff criteria are used to define a hatch region.

**Hatch Table**

Hatch Area

Specifies the upper and lower boundaries for each hatch area. Use the buttons along the top of the table to manage these boundaries. Use the + and – buttons to expand and collapse the list of boundaries for each hatch area heading.

Collapse All

Click to collapse the display of boundaries to show only the top-level headings.

Expand All

Click to expand the display to show all boundaries.

Add Upper Boundary

Click to add an upper boundary to the selected hatch area.

Add Lower Boundary

Click to add a lower boundary to the selected hatch area.

Delete

Click to delete the selected boundary.

Profile

Specifies the surface or layout profile that defines each hatch area boundary. Where only a single surface and single layout profile exist in the profile view, the profiles are automatically assigned to boundaries. Click the arrow beside the profile name to select a different profile.

Shape Style

Specifies the shape style applied to the hatch area. Click the name to select another style or modify the style properties. These styles are found on the Toolspace Settings tab ➤ General ➤ Multipurpose Styles ➤ Shape Styles.

**Related procedures:**

■ Adding Hatch Areas to a Profile View (page 1116)

# Projections Tab (Profile View Properties Dialog Box)

Use this tab to show or hide projected objects in a profile view.

Name

Specifies the names and categories of objects that are available for display in the current profile view. Click to add or remove objects in the profile view.

Style

Specifies the display style of each projected object. Click a single entry to change it, or click <set all> for a category to select a style for all objects in that category.

Elevation Options

Specifies whether the elevation of the projected object is determined by its elevation in plan view (Use Object) or manually controlled in the profile view. Click a single entry to change it, or click <set all> for a category to select an option for all objects in that category.

Elevation Value

Specifies the actual elevation of the projected object. Click <set all> for a category to select an elevation for all objects in that category.

Pick Objects

Click to select objects in the drawing and add them to the profile view.

**Related procedures:**

■ Adding Projected Objects to a Profile View (page 1119)

## Pipe Networks Tab (Profile View Properties Dialog Box)

Use this tab to view and change properties of pipe network parts (pipes and structures) in a profile view. You can change the column widths in the table to make it easier to see the contents of cells.

**NOTE** This tab is displayed only if there are one or more pipe networks in the drawing.

The properties table has the following columns:

Name

Displays the name of the pipe network and associated pipe and structure objects.

Draw

Specifies whether to display the pipe or structure in the profile view. Selected items are displayed in the profile view.

**NOTE** Only checked items can have their Layer, Style, and Style Override modified.

Description

Displays an optional description of the pipe or structure.

Layer

Specifies the drawing layer on which the pipe or structure profile is placed. Click to display Layer Selection dialog box from which you can modify the layer.

Style

Specifies the style used by the pipe or structure. Click to display the Pick Style For Pipe dialog box from which you can modify the style.

Style Override

Specifies whether to override the style for the selected pipe or structure for this profile view. This style is used only in the current profile view. Any other profile view that is plotting the pipe or structure would use the style as specified by the Style column value. Click to select a style in the Pick Style For Pipe dialog box.

Show Only Parts Drawn In The Profile View

Specifies whether to display only the pipes and structures that are selected to draw in the profile view.

**Related procedures:**

- [Displaying Pipe Networks in Profile Views](#) (page 1220)

- [Profile View Properties](#) (page 1024)

# Profile View Style Dialog Box

Use this dialog box to define profile view styles that control the format for titles, axis annotation, and other elements of a profile view.

**See also:**

- [Profile View Styles](#) (page 1029)

# Information Tab (Profile View Style Dialog Box)

Use this tab to record basic information about the profile view style.

Name

Specifies the name of the profile view style.

Description

Specifies an optional description of the profile view style.

Created By

Displays the Windows login name of the person who created the profile view style.

Date Created

Displays the date and time the profile view style was created.

Last Modified By

Displays the Windows login name of the person who last modified the profile view style.

Date Modified

Displays the date and time the profile view style was last modified.

# Graph Tab (Profile View Style Dialog Box)

Use this tab to set the vertical scale and direction of the profile view.

## Vertical Scale

Use the controls in this area to specify how much to increase the elevation values for greater visibility in the profile view. Specify the profile view vertical scale in either of the following ways:

- By Vertical Exaggeration — Enter a value in the Vertical Exaggeration field. The Vertical Scale value is calculated automatically.
  The Current Horizontal Scale (HS) factor of the drawing is divided by the Vertical Scale value (VS) to obtain the profile view Vertical Exaggeration value (VX).

$$\frac{HS}{VS} = VX$$

For example, if the Current Horizontal Scale is 50, and you want the profile view Vertical Scale to be 1"=10′, then set the Vertical Exaggeration to 5.

$$\frac{50}{10} = 5$$

■ By Vertical Scale — Select a scale from the Vertical Scale list. Alternatively, enter a custom value in the Custom field. The Vertical Exaggeration value is calculated automatically.

Vertical Scale

The profile view vertical scale. Select a vertical scale from the list, or enter a custom scale in the Custom Scale field.
Vertical Scale (VS) = Current Horizontal Scale (HS) divided by Vertical Exaggeration (VX)

$$VS = \frac{HS}{VX}$$

Custom Scale

Specifies the current Vertical Scale factor. Enter a value in this field to specify a scale factor that is not provided in the Vertical Scale list.

Current Horizontal Scale

The overall drawing Scale specified in the Drawing Settings dialog box (page 1851). This field is not editable. The horizontal scale must be changed in the Drawing Settings dialog box.

Vertical Exaggeration

Specifies by how much the elevation values increase in the profile view for greater visibility. Enter either 1 for no increase in scale or a larger number to increase the scale. The elevation values are multiplied by this value, so a larger number increases the amount of exaggeration in the profile view.

**Profile View Direction**

 Specifies the direction of the profile in the profile view grid:

■ **Left To Right**: The profile is drawn with 0 or the lowest station number on the left the highest station number on the right.

■ **Right To Left**: The profile is drawn with the highest station number on the left and 0 or lowest station number on the right.

## Grid Tab (Profile View Style Dialog Box)

Use this tab to specify clipping, padding, and axis offset options on the profile view grid.

**Grid Options**

 Clip Vertical Grid

Specifies that vertical grid lines are only drawn below the profile line. If Clip to Highest Profile is not selected, the vertical lines are drawn only below the profile line that has Clip Grid selected on the Profiles tab in the Profile View Properties dialog box (page 2179).
**Clip to Highest Profile(s)**: Specifies that vertical profile view grid lines are drawn to the highest of all displayed profiles.

**Omit Grid in Padding Areas**: Specifies that vertical profile view grid lines are not drawn in the padding areas.

 Clip Horizontal Grid

Specifies that horizontal grid lines are only drawn below the profile line. If Clip to Highest Profile is not selected, the horizontal lines are drawn only below the profile line that has Clip Grid selected on the Profiles tab in the Profile View Properties dialog box (page 2179).

**Clip to Highest Profile(s)**: Specifies that vertical profile view grid lines are drawn to the highest of all displayed profiles.

**Omit Grid in Padding Areas**: Specifies that horizontal profile view grid lines are not drawn in the padding areas.

## Grid Padding

 Grid Padding (Major Grids)

Specifies the distance (in major grid blocks) to add to the extents of the profile view in relation to the profiles it contains.

Above Maximum Elevation

Specifies the number of grid blocks that the profile view extends above the maximum profile elevation. A positive value enables the graph to be taller than the extents of the profile drawn in the profile view.

Below Datum

Specifies the number of grid blocks that the profile view extends below the datum or minimum profile elevation. A positive value enables the graph to extend below the extents of the profile drawn in profile view.

To Left

Specifies the number of major grids to place before the left-most station value of the alignment. A positive value enables the graph to be wider than the extents of the profile drawn in the profile view.

To Right

Specifies the number of major grids to place after the right-most station value of the alignment. A positive value enables the graph to be wider than the extents of the profile drawn in the profile view.

**NOTE** The To Left and To Right settings are not applied if the Station Range is set to User Specified. Set the Station Range option in the Profile View Properties dialog box (page 2178).

## Axis Offset

**NOTE** The axis offset distance is specified in plotted units, which is based on the Drawing Units and Scale settings. Set the Drawing Units and Scale on the Units and Zone tab of the Drawing Settings dialog box (page 1851).

 Axis Offset (Plotted Units)

Specifies the distance (in plotted units) to offset the profile view horizontal and vertical axes from the grid extents (including padding). A positive value creates a blank area between the axis and the grid.

Above Maximum Elevation

   Specifies the distance to offset the horizontal axis above the vertical extent.

Below Datum

   Specifies the distance to offset the horizontal axis below the vertical extent.

To Left

   Specifies the distance to offset the horizontal axis to the left of the horizontal extent.

To Right

   Specifies the distance to offset the horizontal axis to the right of the horizontal extent.

# Title Annotation Tab (Profile View Style Dialog Box)

Use this tab to specify settings for the graph view and axis titles.

## Graph View Title

Text Style

   Specifies the text style for the profile view title.

Text Height

   Specifies the current text height in drawing units.

Title Content

   Specifies the text elements used to compose the title, such as horizontal alignment (HA) name and station

   range. To change title format or content, click ![icon] to open the [Text Component Editor dialog box](#) (page 1961).

## Title Position

These controls specify the position of the profile view title. Use Location and Justification settings for basic positioning, then X and Y offsets for more precise placement, if necessary.

Location

   Specifies the location of the title in relation to the profile view grid: either top, bottom, left, or right.

Justification

   Specifies the justification (alignment) of the title. If Location is top or bottom, justification values are left, right, or center. If Location is right or left, justification values are top, middle, or bottom.

X Offset

   Specifies a horizontal offset from the position set by the Location and Justification values. Enter a positive number to move the title right or a negative number to move it left.

Y Offset

   Specifies a vertical offset from the position set by Location and Justification values. Enter a positive number to move up the title or a negative number to move it down.

Border Around The Title

   Specifies whether a border line is drawn around the title block.

Gap

   Specifies the distance between the title and its border.

## Axis Title Text

Separate titles can be displayed at each of the axes. Use the radio buttons around the graphic to select the axis title to edit. Configure title settings and click Apply. Repeat that process with each axis title.

**NOTE** The Axis Title selections do not control axis title visibility. Visibility controls for all profile view components are located on the Display tab (page 2192).

 Axis Title Text

Specifies which axis title is controlled by the text editing tools.

Title Text

Specifies the title text for the horizontal axis. To change the title content, click  to open the Text Component Editor dialog box (page 1961).

Location

Specifies the location of the title on the axis: either Left, Center, or Right. For more precise control, use the X and Y Offsets.

Text Style

Specifies a text style for use in the horizontal axis title.

Text Height

Specifies the title text height (in plotted units).

Rotation

Specifies the angle for the title. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the title is not rotated.

X Offset

Specifies a horizontal offset from the Location setting. Enter a positive number to move the title right or a negative number to move it left.

Y Offset

Specifies a vertical offset from the Location setting. Enter a positive number to move up the title or a negative number to move it down.

## Horizontal Axes Tab (Profile View Style Dialog Box)

Use this tab to specify the settings for the tick marks on the horizontal axes of the profile view.

 Select Axis to Control

Specifies the axis for which you adjust the Major and Minor Tick Details.

**NOTE** The bottom axis controls the major and minor grid spacing.

Top

Adjust Major and Minor Tick Details along the top axis.

Bottom

Adjust Major and Minor Tick Details along the bottom axis.

Tick and Label Start Station

Specifies whether ticks and labels are placed at the start station on the horizontal axes.

**Major Tick Details**

Interval

Specifies the spacing of major ticks on the horizontal axis, using the actual ground units.

Tick Size

Specifies the height of the tick on the horizontal axis in plotted units.

Tick Justification

Specifies the position of the tick relative to the axis: top, bottom, or center.

Text Height

Specifies the height of text (in plotted units) used to label major ticks on the horizontal axis.

Tick Label Text

Specifies a label property for the tick. To change the text format or content, click ![icon] to display the Text Component Editor dialog box (page 1961).

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

**Minor Tick Details**

Interval

Specifies the spacing of minor ticks on the horizontal axis, using the actual ground units.

Tick Size

Specifies the height of the tick on the horizontal axis in plotted units.

Tick Justification

Specifies the position of the tick relative to the axis: top, bottom, or center.

Text Height

Specifies the height of text (in plotted units) used to label minor ticks on the horizontal axis.

Tick Label Text

Specifies a label property for the tick. To change the text format or content, click ![icon] to display the Text Component Editor dialog box (page 1961).

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

  Specifies a vertical offset for the tick label from the bottom of the tick.

# Vertical Axes Tab (Profile View Style Dialog Box)

Use this tab to specify the settings for the tick marks on the vertical axes of the profile view.

 Select Axis to Control

Specifies the axis to adjust the Major and Minor Tick Details.

**NOTE** The left axis controls the major and minor grid spacing.

Left

  Adjust Major and Minor Tick Details along the left axis.

Right

  Adjust Major and Minor Tick Details along the right axis.

Tick and Label Start Station

  Specifies whether ticks and labels are placed at the start station on the vertical axes.

## Major Tick Details

Interval

  Specifies the spacing of major ticks on the horizontal axis, using the actual ground units.

Tick Size

  Specifies the height of the tick on the horizontal axis in plotted units.

Tick Justification

  Specifies the position of the tick relative to the axis being annotated: left, right, or center.

Text Height

  Specifies the height of text (in plotted units) used to label major ticks on the vertical axis.

Tick Label Text

  Specifies a property with which the tick is labeled. To change the text format or content, click  to display the Text Component Editor dialog box (page 1961).

Text Style

  Specifies the style of the tick label text.

Rotation

  Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

  Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

  Specifies a vertical offset for the tick label from the bottom of the tick.

**Minor Tick Details**

Interval
Specifies the spacing of minor ticks on the vertical axis, using the actual ground units.

Tick Size
Specifies the height of the tick on the vertical axis in plotted units.

Tick Justification
Specifies the position of the tick relative to the axis being annotated: left, right, or center.

Text Height
Specifies the height of text (in plotted units) used to label minor ticks on the vertical axis.

Tick Label Text
Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box (page 1961).

Text Style
Specifies the style of the tick label text.

Rotation
Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset
Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset
Specifies a vertical offset for the tick label from the bottom of the tick.

## Display Tab (Profile View Style Dialog Box)

Use this tab to manage the display of profile view components in 2D and 3D.

For more information about the Component Display section of this tab, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Profile View Style Dialog Box)

Use this tab to review and adjust settings for the profile view style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Project Objects to Profile View Dialog Box

Use this dialog box to specify objects in the current drawing to be projected into a profile view.

To highlight an object in the drawing, select its row in this table.

Name
Displays the objects that can be projected, divided into categories. Select the ones you want to project.

Style
Specifies the current style for each object. You can change the style used to project an object. Select <set all> at the category level to change all objects in that category in one operation.

Elevation Options

Specifies how the elevation of each object is determined. Longitudinal objects, such as feature lines and 3D polylines, have multiple elevation values; their elevation options can differ from those for point objects.

- Use Object: Elevation is read from the object properties.

- Surface: The object is set to the elevation of a selected surface.

- Manual: You can specify the elevation. This option cannot be used with longitudinal objects.

- Profile: The object is set to the elevation of a selected profile.

- <set all>: Used at the category level to specify the elevation option for all projected objects in the category.

Elevation Value

Specifies the elevation of each projected object. You can change the value here for point objects if the elevation option is manual.

Label Style

Specifies the style of label applied to the projected object. Select <none> to display no label. Select <set all> at the category level to set a label style for all projected objects in the category.

Pick Objects

Click to select objects in the drawing and add them to the profile view.

**Related procedures:**

- Adding Projected Objects to a Profile View (page 1119)

## Raise/Lower PVI Elevations Dialog Box

Use this dialog box to change the elevation of an entire profile either for layout profiles or static surface profiles, or a range of stations.

In the Elevation Change field, enter either a positive number to raise the PVIs, or a negative number to lower them. In the PVI Range area, choose one of the following options:

- **All**: To raise or lower the entire profile

- **Station Range**: To raise or lower a subset of PVIs, specified by start and end station values

Default values for the station range are the beginning and end of the profile. If a station equation is in effect, use the station equation values to specify a range.

**Related procedures:**

- Editing PVIs (page 1093)

## Superimpose Profile Options Dialog Box

Use this dialog box to specify the station range of a superimposed profile and the accuracy of curve approximation.

**See also:**

- Creating a Superimposed Profile (page 1054)

## Limits Tab (Superimpose Profile Options Dialog Box)

Use this tab to specify the start and end stations for a superimposed profile.

Specify the station range either by numeric entry or by clicking a station along the parent horizontal alignment.

---

**NOTE** If you want the superimposed profile's station range to remain fixed, then select the Select Start and Select End check boxes. Selecting these options prevents the superimposed profile from reacting to changes in the station range of the source alignment. If the Select Start and Select End boxes are not selected, then the start and end of the superimposed profile is locked to the start and end of the source alignment. If the start or end of the source alignment is then changed, the superimposed profile updates dynamically.

---

## Accuracy Tab (Superimpose Profile Options Dialog Box)

Use this tab to specify the accuracy of curves in the superimposed profile.

Specify the mid-ordinate distances used for approximating horizontal and vertical curves. A small value results in the use of many short line segments to represent a curve. A large value results in longer segments and a coarser curve, with a shorter drawing time.

# Vertical Curve Settings Dialog Box

Use this dialog box to specify the type of curve to be placed between tangents in a profile.

Curves of the type defined here are used if you select the ⛏ Draw Tangents With Curves button on the Profile Layout Tools dialog box (page 2170).

After you select which of the three vertical curve types to use, specify the related parameters for crest curves and sag curves as follows:

■ **Parabolic**: Specify either curve length or K value.

■ **Circular**: Specify either curve length or radius.

■ **Asymmetric**: Specify separate lengths for the parabola in (Length1) and the parabola out (Length2).

Click OK to save the settings.

**Related procedures:**

■ Vertical Curve Design (page 1068)

■ Creating Layout Profiles (page 1048)

# Project Management Dialog Boxes

<div style="text-align: right; font-size: 3em; font-weight: bold;">65</div>

The following topics provide information about the Project Management dialog boxes.

## Add Points To Project Dialog Box

Use this dialog box to create project points from drawing points by adding them to a project.

Point List

Lists the drawing points to be added to the project.

Comment

Specifies a comment to be associated with the point database for this operation. This comment can be viewed in the Autodesk Vault tool by a system administrator.

Check In Options

Specifies additional actions to take after the project points are created.

■ Check In: Creates project points, and then relinquishes your control over the created points, making them available for someone else to edit.

■ Check In And Keep Checked Out: Creates project points and then checks them out to you.

■ Check In And Protect: Creates project points and then protects them, preventing others from editing them.

**Related procedures:**

■ [Adding Points to a Project](#) (page 168)

## Add To Project Wizard

Use this wizard to add a drawing to a project.

This wizard leads you through the steps of adding a drawing to a project.

■ Select the project to which you will add a drawing.

■ Specify the location of the drawing within the project.

- Specify which dependent files will be added to the project, and whether DWF files will be created for those files.

- Specify which drawing objects will be shared across the project.

**Related procedures:**

- [Adding a Drawing to a Project](#) (page 156)

## Select Project Page (Add To Project Wizard)

Use this page to select the project to which you will add a drawing.

The available projects are listed. The active project is the default selection. If you want to select a different project, click it in the list.

Click Next to move to the next page.

## Select Drawing Location Page (Add To Project Wizard)

Use this page to specify the location of the drawing within the project.

The selected project is shown, along with any existing subfolders.

Folder List

Click a subfolder to specify the location of the drawing. Optionally, click  to create a new folder within the project under the currently selected folder.

Click Next to move to the next page.

## File Dependencies Page (Add To Project Wizard)

Use this page to review any dependent files to be added to the project along with the drawing, and to specify the creation of DWF files.

DWF files display the drawing's preview in Prospector and in the Autodesk Vault tool.

Keeps Files Checked Out
Specifies that the files will be added to the project, but the drawing will remain checked out to the working folder, available for revisions. The file remains open in the current AutoCAD Civil session.

Project Files
Specifies which files to add to the project. If you do not want to add a dependent file, clear its check box.

Create DWF
Specifies whether to create a DWF file.

Enter Version Comments
Specifies an optional description of the file you are adding to the project.

Click Next to move to the next page.

## Share Data Page (Add To Project Wizard)

Use this page to specify which objects in the drawing to share across the project.

A shared object is available for use by others. It is listed in the project's Prospector tree and can be referenced by other project drawings.

**NOTE** You should only share objects that are required by others.

Object

Lists the objects in the drawing that can be shared with others. Select the objects that you want to share. If you select an object collection such as Alignments, all objects in the collection are selected.

Status

Displays an error message if an object cannot be shared.

Description

Specifies an optional description of the shared object.

Click Finish to complete the process of adding the drawing to a project.

**Related procedures:**

■ Adding a Drawing to a Project (page 156)

# Check In Drawing Wizard

This wizard leads you through the steps of checking a drawing in to a project.

■ Specify which dependent files will be checked in to the project, and whether DWF files will be created for those files.

■ Specify which drawing objects will be shared across the project.

## File Dependencies Page (Check In Drawing Wizard)

Use this page to choose the dependent files to be checked in to the project along with the drawing.

Keeps Files Checked Out

Specifies that the files will be checked in and the versions will be updated, but the drawing will remain checked out to the working folder and available for revisions.

Project Files

Specifies which files will be checked in to the project. If you do not want to check in a dependent file, clear its check box.

Create DWF

Specifies whether to create a DWF file for the drawing.

Enter Version Comments

Specifies an optional description of the files you are checking in to the project.

Click Next to move to the next dialog box.

## Share Data Page (Check In Drawing Wizard)

Use this page to review the set of objects in the drawing that will be shared, allowing others to reference them in drawings.

If the drawing contains objects that have already been shared, they are not listed.

Object

Lists the objects in the drawing that can be shared with others. Select the objects that you want to share. If you select an object collection such as Alignments, all objects in the collection are selected.

Status

Displays an error message if an object cannot be shared.

Description

Displays a description of the shared object, which you can change.

Click Finish to complete the process of checking in the drawing.

**Related procedures:**

■ Checking a Drawing in to a Project (page 158)

# Check In Points Dialog Box

Use this dialog box to update project points with changes that you have made.

You must check the points in from the same drawing that you checked them out from.

Point list

Lists the project points to be checked in to the project.

Comment

Specifies a comment to be associated with the point database for this operation. This comment can be viewed in the Autodesk Vault tool by a system administrator.

Check In Options

Specifies additional actions to be taken after the points are checked in.

■ Check In: Updates the project points, and then relinquishes your control over them, making them available for someone else to edit.

■ Check In And Keep Checked Out: Updates the project points and keeps them checked out to you.

■ Check In And Protect: Updates the points and then protects them, preventing others from editing them.

**Related procedures:**

■ Checking In Points (page 170)

# Check Out Drawing Dialog Box

Use this dialog box to check out a project drawing.

Include File Dependencies

Specifies whether dependent files should be checked out with the project drawing.

Get Latest Version

Specifies whether local copies of files are overwritten.
Selected: The latest versions in the project database are checked out, overwriting the related files in the working folder.
Cleared: Files in the working folder are not overwritten, but the latest versions in the project database are checked out. Use this option if you have made changes to the local files and you want to check them in to the database as the latest project version.

**IMPORTANT** If you clear this check box with the intention of checking in files from the working folder, make sure that no one else has revised the project versions of the files since you last checked them out. Such changes would be lost when you check in your working version. If needed, a prior version can be restored by an administrator using the Autodesk Vault tool.

File List

Specifies the files and the dependent files, if applicable, that you want to check out.

Enter Version Comments

Specifies an optional description of the file version you are checking out of the project.

**Related procedures:**

■  Checking a Drawing out from a Project (page 157)

■  Checking Out Project Object Source Drawings (page 165)

# Check Out Points Dialog Box

Use this dialog box to check out project points to the active drawing.

Point List

Lists the project points to be checked out.

**Related procedures:**

■  Checking Out Points (page 169)

# Create Alignment Reference Dialog Box

Use this dialog box to create a data reference to an alignment.

Source Alignment

Specifies the alignment from which to create the reference. Click to change the alignment or to display all available alignments.

Site

Specifies a site for the alignment reference. Either select a site from the Site list or click  to select an object in the drawing. The alignment and the object you select in the drawing are associated with the same site.

**NOTE** Accepting the default <None> selection places the alignment in the top-level Alignments collection in Prospector. See Alignment and Site Interaction (page 721) for more information.

Name

Specifies a system-generated name for the alignment reference. To edit the name, click  .

Description

Specifies an optional description of the alignment reference.

Alignment Style

Displays the current style. Click the arrow to display the alignment styles in the drawing.  specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from drawing.  opens the Style Detail dialog box. Preview the style and creation information.

Alignment Layer

Displays the layer on which the alignment reference will be created. To change the layer click  and select a layer.

Alignment Label Set

Lists the alignment label sets in the drawing. You can select a label set from the list. After you select the label set, use the Selection icon to edit the current label set, copy the current label set, or create a new label set. The Style Detail icon previews the current style.

**Related procedures:**

■ Creating an Alignment Reference (page 893)


# Create Data Shortcuts

Use this dialog box to share objects from the current drawing.

Object Table

Select the objects that you want to reference in other drawings.

Hide Already Published Objects

Click to hide all objects in the table that are already shared.

Pick In Drawing

Click to select objects in the drawing rather than in the table.

**Related procedures:**

■ Creating Data Shortcuts (page 131)


# Create Pipe Network Reference Dialog Box

Use this dialog box to create a data reference to a pipe network.

Source Pipe Network

Specifies the pipe network data shortcut from which to create the reference. Click to change the data shortcut or display all available pipe network data shortcuts.

Network Name

Specifies a system-generated name for the pipe network reference. To edit the name, click  .

Network Description

Specifies an optional description of the pipe network reference.

Network Parts List

Specifies the parts list associated with this pipe network reference.

Layers

Displays the layer on which the pipe network reference will be created. To change the layer click ⊗ and select a layer.

Override Reference Surface

If you select this option, each referenced part references the surface specified here.

Override Reference Alignment

If you select this option, each referenced part references the alignment specified here.

---

**NOTE** If neither Override Reference option is selected, each part from the source drawing matches the same-named object (surface or alignment) in the reference drawing. When specific parts reference surfaces or alignments that cannot be found in the reference drawing, the following warning appears in event viewer: **Reference <Object> not found in reference drawing**. In this case, these network parts are created with a value of <none> for the missing reference object.

---

Include Source Labels

If you select this option, labels or label sets are copied with the reference object as they exist in the source drawing and the label style options are grayed out. Otherwise, the label style options are active and the label styles will be created when the referenced parts are added to the drawing.

---

**NOTE** Spanning pipe labels are created in the reference drawing matching their data from the source drawing. They reference the same parts as were specified when the spanning label was created.

---

Structure Label Style and Pipe Label Style

These fields display the current styles for structure and pipe labels. Click the arrow to display the structure and pipe label styles in the drawing. 📝 specifies the style options. Create a new style, copy or edit the current style selection, or pick a style from drawing. 🔍 opens the Style Detail dialog box. Preview the style and creation information.

**Related procedures:**

■   Creating a Pipe Network Reference (page 1199)

# Create Profile Reference Dialog Box

Use this dialog box to create a data reference to a profile.

Source Profile

Specifies the profile to reference. Click to change the profile or to display all available profiles.

Source Alignment

Displays the alignment data shortcut along which to create the profile data shortcut.

---

**NOTE** If the Source Alignment field is set to <None>, you must create an alignment reference before continuing with the profile reference creation. See Creating an Alignment Reference (page 893).

---

Name

Specifies a system-generated name for the profile. To edit the default naming scheme, click 🔖 .

Description

Specifies an optional description of the profile.

Profile Style

Specifies the current style for the profile. Use the standard controls to edit the style or create a new one.

Profile Layer

Displays the drawing layer on which the profile will be drawn. To change the layer, click ⬟ .

Profile Label Set

Specifies the name of the current profile label set. Use standard controls to review or change the set.

**Related procedures:**

■ Creating a Profile by Reference (page 1055)

# Create Surface Reference Dialog Box

Use this dialog box to create a data reference to a surface.

Source Surface

Specifies the complete path to the surface in the project folder.

Surface Layer

Displays the layer on which the surface will be created.

⬟

Click to open the Object Layer dialog box where you can select a different layer for the surface.

The Properties Table section of the dialog box displays varying properties depending on the type of surface selected.

## Information

Name

Specifies the name of the surface.

**NOTE** To name the surface, click its default name and enter a new name, or click the Name Template button and use the Name Template. For more information, see Name Template Dialog Box (page 1806).

Description

Specifies a description of the surface.

Style

Specifies the style for the surface. Click ⋯ to open the Select Surface Style (page 1801) dialog box where you can select a different style.

Render Material

Specifies the render material (page 1577). Click ⋯ to select a render material.

## Grid Parameters

This property group is displayed when a grid surface is selected.

Grid X Spacing

Specifies the X distance between the grid lines. When selected, you can click ▦ to digitize the spacing in the drawing area.

Grid Y Spacing

Specifies the Y distance between the grid lines. When selected, you can click ▦ to digitize the spacing in the drawing area.

Orientation

Specifies the direction for the grid in the X and Y directions. When selected, you can click  to pick two points in the drawing to define the orientation direction.

## Volume Surfaces

Base Surface

Specifies the base surface from which the volume surface is generated. Enter the surface name or click  to open the Select Base Surface (page 1810) dialog box where you can select the surface from the list.

Comparison Surface

Specifies the comparison surface from which the volume surface is generated. Enter the surface name or click  to open the Select Comparison Surface (page 1810) dialog box where you can select the surface from the list.

**Related procedures:**

■ Creating a Surface Reference (page 609)

# Create View Frame Group Reference Dialog Box

Use this dialog box to create a data reference to a view frame group.

## View Frame Group

Source View Frame Group

Specifies the view frame group from which to create the reference. Click to change the group or to display all available view frame groups.

Name

Specifies a system-generated name for the view frame group reference. To edit the name, click  .

Description

Specifies an optional description of the view frame group reference.

## View Frame

View Frame Style

Displays the style used to display the view frames in this view frame group reference.

View Frame Label Style

Displays the style used to display labels on the view frames used in this view frame group reference. For more information, see Understanding Labels (page 1466).

View Frame Layer

Displays the layer on which the view frames used in this view frame group reference will be created. To change the layer click  and select a layer.

## Match Line

Match Line Style

Displays the style used to display match lines and match line mask areas in this view frame group reference.

Match Line Left Label Style

Displays the style used for the labels that are on the left side of a match line in this view frame group reference. For more information, see Understanding Labels (page 1466).

Match Line Right Label Style

Displays the style used for the labels that are on the right side of a match line in this view frame group reference. For more information, see Understanding Labels (page 1466).

Match Line Layer

Displays the layer on which the match lines used in this view frame group reference will be created. To change the layer click ⬟ and select a layer.

Alignment

Specifies the alignment this view frame group reference is associated with.

**Related procedures:**

■ Creating a View Frame Group Reference (page 1709)

# Databases Dialog Box

Use this dialog box to review the list of available databases on a specified Vault server and select the one to log in to.

# Delete Points From Project Dialog Box

Use this dialog box to delete points from a project.

You must check out a point before you can delete it.

Point list

Lists the project points to be deleted.

**Related procedures:**

■ Deleting Project Points (page 172)

# Get Latest Version Dialog Box

Use this dialog box to get a read-only local copy of a project file.

Include File Dependencies

Specifies whether dependent files should be displayed when they exist.

File List

Lists the available files and related project files. Select the files and dependent files you want.

**Related procedures:**

■ Getting the Latest Version of a Drawing (page 159)

■ Getting the Latest Version of Project Objects (page 163)

# Get Points From Project Dialog Box

Use this dialog box to create a read-only local copy of project points in the active drawing.

Point list

Lists the project points to be copied into the active drawing.

**Related procedures:**

■ Copying Project Points into a Drawing (page 171)


# Log In Dialog Box

Use this dialog box to log in to a Vault server and a Vault database.

User Name
 Specifies the user name for logging in to the database. This name must be known to the Vault server.

Password
 Specifies the password associated with the user name.

Server
 Specifies the Vault server to log in to. The list shows all Vault servers that the user has accessed.

Database
 Specifies the database on the Vault server to access. To see the list of available databases, click ⬚ .

Automatically Log In Next Session
 Specifies whether the user is automatically logged in to the above-specified server and database when the next AutoCAD Civil session starts.

**Related procedures:**

■ Logging in to a Vault Server (page 149)


# New Data Shortcut Folder

Use this dialog box to create a new folder for a project that uses data shortcuts.

Working Folder
 Specifies the parent folder in which the project folder will be created. Default value is C:\Civil Projects.

Name
 Enter a name for the new project folder.

Description
 Optionally enter a text description of the project.

Use Project Template
 Select if you want to use a template (standard set of subfolders) for the new project. This option is recommended.

Project Templates Folder
 Displays the available project templates in the current folder. Default value is C:\Civil Project Templates.
 Click ⬚ to browse to a different template folder.

Project Template
 Select the template you want to use for the new project. Default value is _Sample Project.

**Related procedures:**

■ Creating a New Shortcuts Folder (page 129)

# New Folder Dialog Box

Use this dialog box to specify the name of a new folder you are creating within the project folder.

Enter a name and click OK to continue.

# New Project Dialog Box

Use this dialog box to specify the name, description and template when creating a new Vault project.

Name
  Specifies the project name.

Description
  Specifies a description of the project.

## Project Template

Use Project Template
  Select if you want to use a template (standard set of subfolders) for the new project. This option is recommended.

Project Templates Folder

  Specifies the location for your project template. folder. Click  to browse to the folder.

Project Template
  Specifies the name of the project template.

Created By
  Displays the name of the user who created the project.

Date Created
  Displays the date and time the project was created.

**Related procedures:**

■  Creating Projects (page 150)

# Project Properties Dialog Box

Use this dialog box to review the properties of a Vault project.

Name
  Specifies the project name.

Description
  Specifies a description of the project.

Created By
  Displays the name of the user who created the project.

Date Created
  Displays the date and time the project was created.

**Related procedures:**

■  Creating Projects (page 150)

# Properties - Civil Projects Dialog Box

Use this dialog box to review or change the Vault database or the working folder.

Vault Database
   Specifies the database that you are logged in to.

Working Folder
   Specifies the current location for your local copies of files. Click [...] to browse to another folder.

Project Templates Folder
   Specifies the location for your project template folder. Click [...] to browse to the folder.

Current User Name
   Displays the user name for the current session.

**Related procedures:**

■  Changing the Working Folder (page 150)

# Protect Points Dialog Box

Use this dialog box to prevent others from editing project points.

Point list
   Lists the project points to be protected.

**Related procedures:**

■  Protecting Project Points (page 171)

# Reset Points To Checked In Dialog Box

Use this dialog box if you are a system administrator and need to reset checked-out points.

Point list
   Lists the project points to be reset.

**Related procedures:**

■  Resetting Project Points Set by Others (page 174)

# Select Projects To Display Dialog Box

Use this dialog box to select the projects to display in the Prospector tree and the Add to Project (page 2196) dialog box.

The dialog box contains a list of all the projects available in the Vault database you are logged in to. Project names with a selected check box will be displayed in the Projects collection in the Master View of the Prospector tree.

When an active project has been set, the name of the active project is not displayed in the list.

   Selects all projects in the list.

Clears all projects in the list.



Selects multiple project names. Use Ctrl+click or Shift+click to select multiple project names, then click this button to select their check boxes.



Clears multiple project names. Use Ctrl+click or Shift+click to select multiple project names, then click this button to clear their check boxes.

Name

Lists all the projects available in the database. Click the column heading to sort the projects alphabetically by name.

**Related procedures:**

■ Selecting Projects to Display (page 152)

## Set Data Shortcuts Folder

Use this dialog box to select a different project folder as a source of data shortcuts.

Typically you select a different project folder when you want to associate the current drawing with that project.

Folder List

Click a table row, then click OK to select the associated project folder. You can click the Name or Description heading to sort the table entries alphabetically.

Folder Name

Displays the currently selected project folder.

**Related procedures:**

■ Setting the Data Shortcuts Folder (page 130)

## Sync To Project Dialog Box

Use this dialog box to update out-of-date local copies of project objects and project points contained in a drawing.

Object

Lists the local objects in the drawing that are older than the latest project version. Select all objects that you want to synchronize. If you select or clear an object collection, such as Alignments, the action affects all objects in the collection.



Specifies that the object is a reference to a project object.



Specifies that the object is not synchronized with the project data.

**Related procedures:**

■ Synchronizing Drawing Data with Project Data (page 161)

# Undo Check Out Dialog Box

Use this dialog box to reset a checked-out drawing and related files to a checked-in state.

This operation is useful if you decide to make no changes to a checked-out drawing, or if you make some changes but do not want to update the master version of the file with the changes.

Include File Dependencies
  Specifies whether related files are reset to a checked-in state.

File List
  Lists the available files and related project files. Select the files and dependent files, if applicable, that you want to reset to a checked-in state.

Version Comments
  Displays any comment that was added when the drawing was checked out.

**Related procedures:**

■ Resetting a Checked out Drawing (page 160)

■ Resetting a Checked Out Source Drawing (page 165)

# Undo Check Out Points Dialog Box

Use this dialog box to reset your checked-out points to a checked-in state.

Point list
  Lists the project points to be checked in.

**Related procedures:**

■ Resetting Points You Checked Out (page 173)

# Unprotect Points Dialog Box

Use this dialog box to allow others to edit protected project points.

Only the person who protected a project point can unprotect it.

Point list
  Lists the project points to be unprotected.

**Related procedures:**

■ Unprotecting Project Points (page 172)

# Reports Dialog Boxes

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Use the following links to access information about the AutoCAD Civil Reports dialog boxes.

## Toolbox Editor Vista

Use this panorama vista to add custom reports or other custom tools to the Toolbox.

**Expand All Categories**

Expands all categories.

**Collapse All Categories**

Collapses all open categories.

**Add Root Item**

Inserts a new collection at the same level as the Reports Manager.

**Tree Items**

Right-click a tree item to display commands to add and remove categories and tools.

**Properties & Values**

Specifies the values for categories in tools.

For example, if you are adding a new report, you need to specify the Report Type (VBA, COM, .NET, or XSL), the file to use, and the macro to run.

You can optionally specify a help file and help topic to display.

**Reload**

Clears any changes you have made in the current session and reloads the data.

**Save**

Saves any changes you have made but keeps the window open for further edits.

**Dismiss**

Closes the Toolbox Editor vista.

# Edit Report Settings Dialog Box

Use this dialog box to edit settings for all report types.

⊡▼ Expand All Categories

Expands all categories in the dialog box.

⊡▲ Collapse All Categories

Collapses all open categories in the dialog box.

Categories and Settings

Use the +/- boxes to display or hide the settings in the category. Use the Value column to specify a value for a setting.

# Create Reports - PI Station Report

Use this dialog box to select the alignments and specify station ranges for the PI Station report.

This report lists tangent information for a range of alignment stations. No curve data is reported.

### List of Alignments

Include

Specifies the alignments to include in the report. Select a check box to include the alignment.

Name & Description

Displays the name and description of the alignments in the drawing.

Station Start & Station End

Displays the start and end stations for a selected alignment
You can edit these values by highlighting an alignment in the list and adjusting the Report Settings (see below).

### Report Settings

Start Station & End Station

Displays the start and end stations for a highlighted alignment. Click an alignment name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Incremental Station Report

Use this dialog box to select the alignments and specify station ranges for the Incremental Station report.

This report lists northing, easting, and tangent information for a range of alignment stations, based on a specified station increment.

**List of Alignments**

Include

Specifies the alignments to include in the report. Select a check box to include the alignment.

Name & Description

Displays the name and description of the alignments in the drawing.

Station Start & Station End

Displays the start and end stations for a selected alignment
You can edit these values by highlighting an alignment in the list and adjusting the Report Settings (see below).

**Report Settings**

Start Station & End Station

Displays the start and end stations for a highlighted alignment. Click an alignment name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Station Increment

Specifies the station increment to use. For example, specify 50 to report at increments of 50 feet or meters along the alignment.

---

**NOTE** The increment is applied to all alignments. Station 0+00 is always included in the report. Odd stations are not included if they do not match the station increment. For example, if the increment is 50, and the alignment starts at station -10+00, the report will begin at station 0+00. However, if the alignment begins at station -60+00, the report will begin at station -50+00.

---

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Stakeout Alignment Report

Use this dialog box to select the alignments and specify station ranges for an Alignment Stakeout report.

This report lists angle, distance, northing, and easting from the occupied point to each alignment station.

**List of Alignments**

Include

Specifies the alignments to include in the report. Select a check box to include the alignment.

Name & Description

Displays the name and description of the alignments in the drawing.

Station Start & Station End

Displays the start and end stations for a selected alignment
You can edit these values by highlighting an alignment in the list and adjusting the Report Settings (see below).

**Stakeout Options**

Point Occupied

Specifies the point from which the angles to the alignment are calculated. You can enter a point number

or click  to select a point in the drawing.

Backsight Point

Specifies the backsight point. The backsight point and occupied point cannot be the same point.

Select Angle Type

Specifies the stakeout angle type to use.

- **Turned +**: Turned angle right

- **Turned -**: Turned angle left

- **Deflect +**: Deflection angle right

- **Deflect -**: Deflection angle left

- **Direction**: Uses azimuths or bearings

backsight point      occupied point

turned angle right



occupied point      backsight point

turned angle left



occupied point

backsight point

deflected angle right



occupied point

backsight point

deflected angle left

## Report Settings

Start Station & End Station

Displays the start and end stations for a highlighted alignment. Click an alignment name in the list to highlight it.

You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Station Increment

Specifies the station increment to use. For example, specify 50 to report at increments of 50 feet or meters along the alignment.

**NOTE** The increment is applied to all alignments. Station 0+00 is always included in the report. Odd stations are not included if they do not match the station increment. For example, if the increment is 50, and the alignment starts at station -10+00, the report will begin at station 0+00. However, if the alignment begins at station -60+00, the report will begin at station -50+00.

Offset

Specifies a distance from the alignment the stakeout report will use in calculations. Enter a negative number to indicate a left offset.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Alignment Design Criteria Verification

Use this dialog box to compare alignment parameters with specified design criteria.

This report lists displays each entity's parameters, and whether the entity clears or violates the specified design criteria and design checks.

For more information, see Criteria-Based Alignment Design (page 866).

### List of Alignments

Include

Specifies the alignments to include in the report. Select a check box to include the alignment.

Name & Description

Displays the name and description of the alignments in the drawing.

Station Start & Station End

Displays the start and end stations for a selected alignment.
You can edit these values by highlighting an alignment in the list and adjusting the Report Settings (see below).

### Report Settings

Start Station & End Station

Displays the start and end stations for a highlighted alignment. Click an alignment name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Profile Design Criteria Verification

Use this dialog box to compare profile parameters with specified design criteria.

This report lists displays each entity's parameters, and whether the entity clears or violates the specified design criteria and design checks.

For more information, see Criteria-Based Profile Design (page 1044).

**List of Profiles**

Include

Specifies the profiles to include in the report. Select a check box to include the profile.

Name & Description

Displays the name and description of the profiles in the drawing.

Station Start & Station End

Displays the start and end stations for a selected profile.
You can edit these values by highlighting a profile in the list and adjusting the Report Settings (see below).

Alignment

Displays the name of each profile's parent alignment.

**Report Settings**

Start Station & End Station

Displays the start and end stations for a highlighted profile. Click a profile name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - PVI Station and Curve Report

Use this dialog box to select the profiles and specify station ranges for a profile PVI station and curve report.

This report lists station, elevation, and grade out for each PVI in an profile. Vertical curve information is included for finished ground profiles.

**List of Profiles**

Include

Specifies the profiles to include in the report. Select a check box to include the profile.

Name & Description

Displays the name and description of the profiles in the drawing.

Station Start & Station End

Displays the start and end stations for a selected profile.
You can edit these values by highlighting a profile in the list and adjusting the Report Settings (see below).

Alignment

Displays the horizontal alignment that the profile is associated with.

**Report Settings**

Start Station & End Station

Displays the start and end stations for a highlighted profile. Click a profile name in the list to highlight it.

You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

# Create Reports - Profile Incremental Station Report

Use this dialog box to select the profiles and specify station ranges for a profile incremental station report.

This report lists PVI stations and elevations of the selected profile at regular intervals, at critical geometry points, and crest and sag curve points.

## List of Profiles

Include

Specifies the profiles to include in the report. Select a check box to include the profile.

Name & Description

Displays the name and description of the profiles in the drawing.

Station Start & Station End

Displays the start and end stations for a selected profile.
You can edit these values by highlighting a profile in the list and adjusting the Report Settings (see below).

Alignment

Displays the horizontal alignment that the profile is associated with.

## Report Settings

Start Station & End Station

Displays the start and end stations for a highlighted profile. Click a profile name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Station Increment

Specifies the station increment to use. For example, specify 50 to report at increments of 50 feet or meters along the profile.

**NOTE** The start and end stations are always included in the report. The increment starts at the beginning station, even if it is an odd value. The increment is applied to all profiles.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Vertical Curve Report

Use this dialog box to select the profiles and specify station ranges for a profile vertical curve report.

**List of Profiles**

Include

Specifies the profiles to include in the report. Select a check box to include the profile.

Name & Description

Displays the name and description of the profiles in the drawing.

Station Start & Station End

Displays the start and end stations for a selected profile.
You can edit these values by highlighting a profile in the list and adjusting the Report Settings (see below).

Alignment

Displays the horizontal alignment that the profile is associated with.

**Report Settings**

Start Station & End Station

Displays the start and end stations for a highlighted profile. Click a profile name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - Slope Stake Report

Use this dialog box to select the corridors and specify sample line groups for a cross section slope stake report. This report identifies the points of slope change for a selected corridor link.

## Select Report Components

Select Corridor

Specifies the corridors to include in the report. Only corridors that are based on alignments with defined sample line groups are displayed.

Select a corridor from the list or click  to select a corridor from the drawing.

Select Alignment

Specifies the alignment to include in the report. Only the alignments contained in the selected corridor are displayed.

Select Sample Line Group

Specifies the sample line group to include in the report. Only the sample line groups for the selected alignment are displayed.

Select Corridor Link

Specifies which corridor link to use in the report. A link is defined as a single straight-line segment between endpoints on a subassembly.

Add

Adds the selected corridor, alignment, and sample line group to the List of Corridors to include in the report.

## List of Corridors

Include

Lists the corridors to include in the report.

**NOTE** When a corridor is highlighted, the start and end stations are editable. See Report Settings below.

Alignment & Sample Line Group

Lists the alignment and sample line group that you selected.

Station Start & Station End

Displays the start and end stations for the selected sample line group.
You can edit these values using the Report Settings (see below).

Delete

Removes the corridor from the list.

## Report Settings

Start Station & End Station

Displays the start and end stations for a highlighted sample line group. Click a sample line group name in the list to highlight it.
You can edit these values to change the range of stations included in the report. You can enter formatted or unformatted values, such as 2+50.95 or 250.95.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

## Report Results

The slope stake report has data formatted in columns. The end columns contain information about the last two offsets in the corridor. The other columns contain information about the point codes.

End Column Descriptions

- **Row 1**: Reports either cut (C) or fill (F) which is determined by the elevational difference between the last two offsets reported.

- **Row 2**: Reports the distance difference between the last two offsets reported.

- **Row 3**: Reports the slope value from the second-to-last offset point to the last offset point.

   For example:
   C 8.47
   @33.87
   S 1:4.00

Report Column Descriptions

- **Row 1**: Reports the point code.

- **Row 2**: Reports the offset value relative to 0+00.

- **Row 3**: Reports the elevation value at that offset.

- **Row 4**: Reports the slope from that point headed toward the next point (away from 0+00).

For example:
Daylight
-110.369
629.991
1:-4.00

**NOTE** For slope values greater than 2:1, the value is reported as 1:X. For slope values less than 2:1, the value is reported as X%.

# Create Reports - Map Check Report

Use this dialog box to select the parcels or survey figures to include in a map check report. This report displays the directions and distances, start and end coordinates, curve data, perimeter, and area, and calculates the error of closure.

## List of Objects

Parcels

Select this option to report on parcels in the drawing. When you select this option, the parcels in the current drawing are listed in the table below.

Survey Figures

Select this option to report on survey figures.

**NOTE** The Survey database in the drawing must be open to report on figures.

Include

Select the individual parcels or figures to include in the report by selecting the check boxes. Use this option to select an individual parcel to change the Point of Beginning (see below).

Select All

Selects all parcels or figures for the report.

Deselect All

Clears the current selection of parcels or figures.

## Analysis

**NOTE** All options in the Analysis section are available for parcel reports only and are unavailable if you select the Survey Figures option.

Point Of Beginning

If only one parcel is selected in the list above, its Point of Beginning X,Y values are displayed here. You can click the icon to specify a different point of beginning to use in the report.

**NOTE** The Point of Beginning (POB) X,Y values are not displayed if more than one parcel is selected in the list above. If you change the POB for one parcel, the change is not saved if you select another parcel in the list. Changes are not saved to the parcel in the drawing.

Process Segment Order Counterclockwise

■ **Selected**: Segments are processed in counter-clockwise order.

■ **Cleared**: Segments are processed in clockwise order.

Enable Mapcheck Across Chord

■ **Selected**: The mapcheck traverse is calculated for curve segments using their chord length.

■ **Cleared**: The mapcheck traverse is calculated for curve segments using their curve length.

Truncate Area (Square Unit)

- **Selected**: The area values output are limited to the specified number of decimal places in the Parcel Area Precision settings without rounding. For example, if the area of a parcel is 43,559.989 square feet and the Area precision is set to 2, then the value will be truncated to 43,559. 98 (it will not be rounded up).

- **Cleared**: The area values output are rounded based on the Parcel Area Rounding settings.

### Settings

Parcel Settings

Displays read-only settings for parcels, as specified in the Parcel settings (page 824).

Figure Settings

Displays read-only settings for survey figures, as specified in the Survey settings (page 225).

### Report Settings

Save Report To

Specifies the name and location of the resulting report.

# Create Reports - Parcel Volume Report

Use this dialog box to select the surface and parcels for a parcel volume report. This report calculates the volume of an area bounded by the selected parcels and is based on an existing volume surface.

### List of Parcels

Parcel

Specifies the parcels to use in the report. Select the check boxes next to the parcels to include.

### Report Settings

Volume Surface

Specifies the volume surface to use in the report. For more information about creating a volume surface, see Calculating Surface Volumes (page 690).

Fill Correction

Specifies the contraction or shrinkage of the fill material. The fill factor can be used to compute the additional material volume requirements.
A factor of 1.00 applies no adjustment to the raw volume.

Cut Correction

Specifies the expansion or swell of the cut material. The volume of material generally expands after removing it. Therefore, the cut factor is usually set to greater than 1, which indicates swell or expansion of the material. For example, a 1.2 cut factor would mean that for every 1 cubic meter of material removed, 1.2 cubic meters of volume would need to be accounted for transport.

Elevation Tolerance

Specifies the difference in elevation that must exist between the two surfaces of the volume surface in order for the volume to be included in the report. The surfaces are compared at each grid node. If the difference in elevation is less than the tolerance value, then it is considered to be zero volume.

**NOTE** This control is available only if the selected Volume Surface is a grid surface.

Save Report To

Specifies the name, file type, and location of the resulting report. Click the folder icon to specify a file type and location.

Create Report
    Generates the report and saves it to the specified location.

Done
    Closes the dialog box.

# Create Reports - Station Offset to Points Report

Use this dialog box to select the points and alignment to use for creating a station offset to points report. This report lists the station and offset of the points relative to the selected alignment.

## List of Points

Select All
    Selects all the points in the drawing. When a point is selected the Include box is checked.

Deselect All
    Deselects all the points in the drawing.

Include
    When selected, includes the point in the report.

## Report Settings

Select Alignment

Specifies the alignment to include in the report. Select an alignment from the list or click  to select an alignment from the drawing.

Save Report To
    Specifies the name and location of the resulting report.

Create Report
    Generates the report and saves it to the specified location.

Done
    Closes the dialog box and discards changes to station ranges.

# Create Reports - HEC-RAS Geographical Data Report

Use this dialog box to select a sample line group and to define reaches to include in a HEC-RAS geographical data report. This report describes the properties of a stream profile and its reaches and includes the cross sectional data along the entire profile.

### Prerequisites

To run this report, the main river must be defined as a horizontal alignment, and sample lines must be cut at all critical points along the river (alignment). To create sample lines, a surface is required.

You must define alignment PIs at stream junctions so you can define reaches for the report. For more information about adding PIs to an alignment, see Alignment Layout Tools (page 1747).

Reaches are defined by selecting station values that represent the beginning and end of the reach.

### Select Sample Line Group

Select Alignment
    Specifies the alignment (river) to include in the report. Only those alignments that have valid sample line groups are listed.

Select Sample Line Group

Specifies the sample line group to include in the report. Only the sample line groups for the selected alignment are displayed.

**Define Reach**

Name

Specifies the name for the reach.

Reach Begin Point

Specifies the beginning point for the reach. The list displays the station values of each PI along the alignment. See Prerequisites (page 2223) above for information on how to add points.

Reach End Point

Specifies the end point for the reach.

Add

Adds the reach to the List Of Reaches to be included in the report.

**List Of Reaches**

Reaches

Each defined reach is displayed in this list. Select a reach and click the Delete button to remove it from the report.

Save Report To

Specifies the name and location of the resulting report.

Create Report

Generates the report and saves it to the specified location.

Done

Closes the dialog box and discards changes to station ranges.

# Create Reports - LandXML Reports

Double-click a LandXML report to launch the Export To LandXML dialog box (page 1983) and then select the data to include in the report.

Each report type filters out irrelevant data when the report is created. If you are creating a points report, for example, only the points that are selected are used in the report. You can clear the selection of other objects in the Export To LandXML dialog box, but that is not necessary.

# Create Reports - Corridor Feature Line Report

Use this dialog box to select a corridor, an alignment, and the feature lines to include in a corridor feature line report. This report displays northing, easting, and elevation coordinates at the specified station interval for the feature lines. It also includes the coordinates of the roadway crown at those stations.

Select Corridor

Specifies the corridor to use in the report.

Select Alignment

Specifies the alignment to use in the report.

Sample Line Group

Specifies the sample line group to include in the report. This list is not available if Corridor Points is selected.

Type selector

■ **Corridor Points**: Reports northing, easting, and elevation coordinates at the specified station interval for the selected feature lines and at the roadway crown.

■ **Corridor Links**: Reports offset, elevations and point code information at each grade break of the selected links.

■ **Surfaces**: Reports a series of data for each station for the selected surface sections.

List of Feature Lines/List of Corridor Links/List of Surfaces

This list changes depending on whether you choose to report the data based on Corridor Points, Corridor Links, or Surfaces. Select individual items, or use the buttons on the right to select all items in the list or to clear the entire selection.

Start Station

Specifies the start station for the report. This station is used for all selected items.

End Station

Specifies the end station for the report. This station is used for all selected items.

Station Interval

Specifies the station interval to use.

Save Report To

Specifies the location and file type for the report. Click the button and specify the folder and file type.

Create Report

Creates the report in the location specified and displays the report results.

When you click Create Report, you may be prompted to add stations to the corridor if the station interval you specified is different than the existing corridor sampling. If you click OK, the command applies the Station Interval setting to the corridor, which can result in increasing the frequency that data is reported. If you click Cancel, only the stations that are common based on the specified interval and the assembly insertion locations are presented in the report.

Increasing the frequency that data is reported is similar to using the Frequency To Apply Assemblies dialog box to add more stations to the corridor. For more information, see To add stations to a region (page 1338). Additional station interval lines are added to the corridor in the drawing when you increase the frequency. You can control the display properties of these lines by changing the Link Style on the Codes tab of the Corridor Properties dialog box. For more information, see Codes Tab (Corridor Properties Dialog Box) (page 1817).

# Create Reports - Lane Slope Report

Use this dialog box to select a sample line group to include in a lane slope report. This report displays the station values for the sample lines associated with the selected corridor and the X,Y coordinates of the sample line at the intersection of the alignment. The report also displays the elevation values of the existing ground and finished ground profiles at those X,Y points and the left and right slopes of the lanes defined between the alignment and the edge of pavement.

List of Sample Line Groups

Specifies the sample line group to include in the report.

Start Station

Specifies the start station for the report. The Start Station and End Station fields are enabled only if one of the sample line groups is selected in the list.

End Station

Specifies the end station for the report. The Start Station and End Station fields are enabled only if one of the sample line groups is selected in the list.

Save Report To

Specifies the location and file type for the report. Click the button and specify the folder and file type.

Create Report

Creates the report in the location specified and displays the report results.

# Create Reports - Daylight Line Report

Use this dialog box to select a sample line group to include in a daylight line report. This report includes the X,Y, and elevation data for the daylight points at the ends of the sample lines. The report also includes the widths of the sample lines to the left and right of the alignment.

List of Sample Line Groups

Specifies the sample line group to include in the report.

Start Station

Specifies the start station for the report. The Start Station and End Station fields are enabled only if one of the sample line groups is selected in the list.

End Station

Specifies the end station for the report. The Start Station and End Station fields are enabled only if one of the sample line groups is selected in the list.

Save Report To

Specifies the location and file type for the report. Click the button and specify the folder and file type.

Create Report

Creates the report in the location specified and displays the report results.

# Create Reports - Incremental Station Elevation Difference Report

Use this dialog box to select a profile to include in an incremental station elevation difference report. This report compares an existing ground profile with a design profile and lists the station, existing ground elevation, design elevation, and the elevational differences between PVIs.

List of Design Profiles

Specifies the design (layout) profile. You must select one. Only dynamic profiles are included in the list. When you click on the design profile, the Start Station and End Station fields become active so you can specify the station range for the report.

List of Existing Profiles

Specifies the existing ground profile. You must select one.

Start Station

Specifies the start station for the report. The Start Station and End Station fields are enabled only if one of the design profiles is selected in the list.

End Station

Specifies the end station for the report. The Start Station and End Station fields are enabled only if one of the design profiles is selected in the list.

Station Interval

Specifies the station interval to use when the Regular Interval option is selected.

**Regular Interval**

Specifies that regular station intervals are included in the report. The interval is based on the Station Interval setting.

**Horizontal Tangent Points**

Specifies that tangent points in the horizontal alignment design are included in the report. For example, PT or PC points.

**Vertical Tangent Points**

Specifies that tangent points in the vertical profile design are included in the report. For example, the start vertical tangent point.

**Existing Ground Points**

Specifies that existing ground grade break points are included in the report.

**Save Report To**

Specifies the location and file type for the report. Click the button and specify the folder and file type.

**Create Report**

Creates the report in the location specified and displays the report results.

# Sections Dialog Boxes

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The following topics provide information about the Sample Line, Section, and Section View dialog boxes.

## Corridor Section Properties Dialog Box

Use this dialog box to edit properties of the selected corridor section, such as its name, section data, and code set styles.

**See also:**

■ Editing Corridor Section Properties (page 1151)

## Information Tab (Corridor Section Properties Dialog Box)

Use this tab to edit primary information about the selected corridor section.

Name

Specifies the name of the corridor section. The default format is <SLG>-n - <SL>-n - <corridor>(n). <SLG> is the parent sample line group name, <SL> is the sample line name, and <corridor> is the name of the corridor on which the corridor section is drawn.
n is an integer greater than or equal to one, which increments as each feature is created in the drawing.

Description

Specifies the optional description of the corridor section.

Object Style

Specifies the name of the style used by the corridor section.

**NOTE** Corridor section object styles are the same as code set styles.



Creates, copies, or edits a style using the Code Set Style dialog box (page 1783). Click the down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.

Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Section Data Tab (Corridor Section Properties Dialog Box)

Use this tab to edit properties of the selected corridor section object.

Description

Specifies the optional description of the current corridor section.

Type

Displays a , which indicates that the section was extracted from a corridor.

Data Source

Displays the name of the data source from which the current corridor section was sampled.

Update Mode

Specifies the type of update mode for the corridor section. Select Dynamic to specify that the corridor section data updates dynamically if the position of the sample line associated with the corridor section changes or the data source (surface or corridor) geometry changes.

Layer

Specifies the drawing layer that contains the current corridor section associated with the selected sample line group. Click in the Layer column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the name of the style of the corridor section. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection.

Station

Displays the station value of the sample line along which the corridor section is sampled.

Left Offset

Displays the left offset value of the current section.

Right Offset

Displays the right offset value of the current section.

Minimum Elevation

Displays the minimum elevation of the section associated with the selected sample line group.

Maximum Elevation

Displays the maximum elevation of the section associated with the selected sample line group.

## Codes Tab (Corridor Section Properties Dialog Box)

Use this tab to display codes and code set style information of the selected corridor section object.

**NOTE** The codes displayed in this tab are as collected from the current corridor section's subassemblies. The Description, Style, and Label Style are as assigned through the associated code set style. To change the display or label styles for any of the codes, edit the object style associated with the current corridor section in the Information tab (page 2229).

Use code set styles to control the visual appearance of a set of codes (link, point, and shape codes) associated with a current corridor section.

Name

Displays an expandable tree with a collection for each type of code: Link, Point, and Shape. Expand the collections to display the individual codes.

Description

Displays the description of each code.

Style

Displays the style assigned to each code.

Label Style

Displays the label style assigned to each code or <none> if no label style is assigned.

Render Material

Displays the render material assigned to each code or <none> if no render material is assigned.

Material Area Fill Style

Displays the Material Area Fill style assigned to each code or <none> if no fill style is assigned.

# Create/Edit Sample Line Group Dialog Box

Use this dialog box to create and edit a sample line group.

**NOTE** Sample lines must be part of one, and only one, sample line group. If a sample line group already exists, the dialog box changes from create mode to edit mode.

Name

Specifies the name of the sample line group. The default format is <SLG>-<[Next Counter]>.
<SLG> is the sample line group name.
<[Next Counter]> is an integer greater than or equal to one, which increments as sample line groups are created in the drawing. This naming convention is based on the name template.
Enter a name or edit a default naming convention in the Name Template dialog box.



Opens the Name Template dialog box (page 1806). Edit the default sample line group naming convention.

Description

Specifies the optional description of the sample line group.

Alignment

Displays the parent alignment under which the sample line group belongs.

Sample Line Style

Specifies the sample line style. To edit a style or create a new one, click  . Select a style from the list of operations. For more information, see Select Style dialog box (page 1805) and Section Data Band Style dialog box (page 2267).

To examine the details of an existing style, select the style name in the list. Click  .
For more information, see the Style Detail dialog box (page 1804).

Sample Line Label Style

Specifies the sample line label style. Standard buttons are also provided to create or edit a label style.

Sample Line Layer

Specifies the drawing layer for the sample line. Standard buttons are also provided to review or change the style.

**Select Data Sources To Sample**

Type

Displays the type of data source from which the section is sampled such as , which indicates a TIN surface; , which indicates a corridor; , which indicates a corridor surface; or  which indicates a pipe network.

Data Source

Displays the name of the data source.

Sample

Specifies a data source from which to sample elevations. A new section object is created for each data source selected. The section object is located at each sample line in the current sample line group.

Style

Specifies the style to apply for each section. Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

**NOTE** If the data source is a corridor, then the style list is for a code set style.

Section Layer

Specifies the drawing layer for the section. Standard buttons are also provided to review or change the style.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface or corridor) geometry changes.

**Related procedures:**

■ Creating Sample Lines (page 1142)

# Create Multiple Section Views Wizard

Use this wizard to create multiple section views.

The multiple section views are created from existing sections, which represent sectional data at the stations along the alignment. You can also display data bands above or below each of the multiple section views.

After configuring the settings in this wizard, click Create Section Views. Specify the location of the lower left corner of the multiple section views grid.

**NOTE** Some controls on this wizard are not available when requisite objects are not present. For example, the Section Views Tables page of this wizard is not available if there is no materials list for the drawing. For information about how to create a material list, see Generating Material Lists (page 1163).

**Related procedures:**

■ Creating/Editing Section Views (page 1153)

# General Page (Create Multiple Section Views Wizard)

Use this page to specify basic information about the section views, including the parent alignment and sample line group name, range of stations, description, style, and layer.

Select Alignment

Specifies the name of the parent horizontal alignment. Select a name from the list or click  to pick an alignment from the drawing.

Sample Line Group Name

Specifies the sample line group from which the section views are created. Select a sample line group from the list or click  to pick a sample line group from the drawing.

Automatic

Specifies that the offset range is set automatically. The start and end stations of the specified sample line group are displayed in the fields next to the selection.

User Specified

Specifies that the station range is set by the user. Specify start and end stations for the multiple station views in the fields next to the selection.

Section View Name

Specifies an editable, system-generated name. The name must remain unique within the drawing.

Description

Specifies an optional description of the section views.

Section View Layer

Specifies the drawing layer for the section views. To review or change layer data, click  .

Section View Style

Specifies the style of the section views. Use standard buttons to review or change the style.

Group Plot Style

Specifies the group plot style for the section views. Use standard buttons to review or change the style.

# Offset Range Page (Create Multiple Section Views Wizard)

Use this page to specify the offset range to which the section views are drawn.

Automatic

Specifies that the offset range is set automatically. The lengths of the left and right swath widths are displayed in the fields next to the selection.

User Specified

Specifies that the offset range is set by the user. Specify the length of the left and right swath widths in the fields next to the selection.

Left

Specifies the offset length of the left swath width.

Right

Specifies the offset length of the right swath width.

# Elevation Range Page (Create Multiple Section Views Wizard)

Use this page to specify the elevation properties of the selected section view.

### Elevation Range

Automatic

Specifies that the elevation range is set automatically. The minimum and maximum heights are displayed in the fields next to the selection.

User Specified

Specifies that the elevation range is set by the user. Specify how the elevation is applied by selecting a Section Views Height Option.

Height

Specifies the height of the elevation.

### Section Views Height Option

From Lowest Elevations Of All Sections

Specifies that the elevation is set from the lowest elevation of each section.

From Mean Elevations Of All Sections

Specifies that the elevation is set from the mean elevation of each section.

Follow A Section

Specifies that the elevation is set from a particular section. Select the section from the Select Section list.

# Section Display Options Page (Create Multiple Section Views Wizard)

Use this page to specify section display options.

### Select Sections To Draw

Name

Displays the names of the sections.

Draw

Specifies whether the sections are drawn in the section views. Select the check box to draw the sections in the section views.

Clip Grid

Specifies whether the grid lines in the current section views are clipped to the sections (horizontal and/or vertical depending on the section view style definition). Select the radio button to clip the grid lines in the section views to one of the named sections.

Change Labels

Specifies the style set from which the section labels will be drawn. Click in this column to open the Select Style dialog box (page 1805).

Style

Specifies the section style. Click in this column to open the Select Section Style dialog box.

Override Style

Specifies the style for the sections. Select the check box to open the Select Section Style dialog box.

# Data Bands Page (Create Multiple Section Views Wizard)

Use this page to specify the properties of the data bands associated with the section view.

Select Band Set

Specifies the set of band styles for the section views. Provides standard controls that you use to review or change the set.

**List of Bands**

Location

Specifies either the top or bottom of the section views. The table below this field shows the current location of the bands in the current band style set.

Band Type

Specifies the data band type, either Section Data or Section Segment.

Style

Specifies the style for the band type.

Surface 1

Specifies a surface sampled by the sample line that supplies the data for the band, including any corridor surface.

Surface 2

Specifies an additional surface sampled by the sample line that supplies the data for the band, including any corridor surface. Not used for Section Segment type data bands.

## Section View Tables Page (Create Multiple Section View Wizard)

Use this page to set volume table properties for the section view.

**NOTE** This page is not available if there is no material list for the drawing. For information on how to create a material list, see Generating Material Lists (page 1163).

Use the following controls to specify the volume table types to draw:

Type

Specifies the type of the volume table(s).

Select Table Style

Specifies the style of the volume table(s). Use the standard to create or edit a label style.

Add >>

Adds a volume table to the list.

**List Of Volume Tables**

Table Type

Displays the current table type.

Style

Specifies the name of the table style that was added using the Select Table Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Material List

Specifies the material list for the volume table.

Materials

Click 🔲 to specify the materials for the volume table.

Layer

Specifies the drawing layer that contains the sampled source.

Split

Specifies whether the layer is split.

Gap

Specifies the distance, in plotted units, between the current data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

Reactivity Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically when either the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

### Position Of Table(s) Relative To Section Views

Section View Anchor

Specifies the section view anchor (Top/Middle/Bottom, Left/Center/Right).

Table Anchor

Specifies the location of the table anchor (Top/Middle/Bottom, Left/Center/Right).

Table Layout

Specifies whether the layout of the tables is horizontal or vertical, in relation to the section view.

X Offset

Specifies the distance, in plotted units, to move the title away from the position set by the table anchor. Enter a positive number to move the title right, or a negative number to move it left.

Y Offset

Specifies the distance, in plotted units, to move the title away from the position set by the table anchor. Enter a positive number to move the title up, or a negative number to move it down.

# Create Sample Lines - By Station Range Dialog Box

Use this dialog box to create sample lines along an alignment for a specified range of stations.

### General

Alignment

Displays the name of the currently selected alignment associated with the range of stations.

### Station Range

From Alignment Start

Specifies whether the station range starts at the beginning of the alignment (that is, at the station located at the beginning of the alignment):

- **True**: Begins the station range at the start of the alignment.

- **False**: Specifies the beginning station in the range. Enter or pick a beginning station.

Start Station

Specifies the beginning station in the range. Enter a station value directly or click 🔖 to pick from the drawing. Available if From Alignment Start is False.

To Alignment End

Specifies whether the station range ends at the end of the alignment (that is, at the final station along the alignment):

■ **True**: Ends the station range at the end of the alignment.

■ **False**: Specifies the ending station in the range. Enter or pick an editing station.

End Station

Specifies the end station in the range. Enter a station value directly or click ⬚ to pick from the drawing. Available if To Alignment End is False.

## Left Swath Width

Snap To An Alignment

Specifies that the sample line snaps the swath widths to an offset alignment at each station:

■ **True**: Snaps the widths to an offset alignment at each station.

■ **False**: Does not snap the widths.

Alignment

Specifies the currently selected alignment. Available if Snap To Alignment is True.

NOTE Although all alignments in the current drawing are listed, do not select the parent alignment. The resulting length will be zero.

Width

Specifies the swath width to the left of the alignment, using the actual ground units. Enter a positive value in ground units. Available if Snap To Alignment is False.

## Right Swath Width

Snap To An Alignment

Specifies that the sample line snaps the swath widths to an offset alignment at each station:

■ **True**: Snaps the widths to an offset alignment at each station.

■ **False**: Does not snap the widths.

Alignment

Specifies the currently selected alignment. Available if Snap To Alignment is True.

NOTE Although all alignments in the current drawing are listed, do not select the parent alignment. The resulting length will be zero.

Width

Specifies the swath width to the right of the alignment, using the actual ground units. Enter a positive value in ground units. Available if Snap To Alignment is False.

## Sampling Increments

Use Sampling Increments

Specifies whether sample lines are created at incremental stations along the alignment:

■ **True**: Changes the default sample increments for either tangents, curves, or spirals. For example, you may want to decrease the values for curves and/or spirals.

■ **False**: Sample lines are not created at stations along the alignment.

Increment Along Tangents

Specifies the sample increment distance along tangents, using the actual ground units. Enter a positive value in ground units.

Increment Along Curves

Specifies the sample increment distance along curves, using the actual ground units. Enter a positive value in ground units.

Increment Along Spirals

Specifies the sample increment distance along spirals, using the actual ground units. Enter a positive value in ground units.

---

**NOTE** Changing sampling increments can affect performance on rebuild. A shorter increment means more sampling stations, and thus a larger drawing.

---

**Additional Sample Controls**

At Range Start

Specifies whether a sample line is created at the beginning of the specified range.

At Range End

Specifies whether a sample line is created at the end of the specified range.

At Horizontal Geometry Points

Specifies whether additional sample lines are created at horizontal geometry points, such as, at the start of a spiral, or at a spiral-curve point.

At Superelevation Critical Stations

Specifies whether additional sample lines are created at superelevation critical stations.

**Related procedures:**

■ Creating Sample Lines (page 1142)

■ Editing Sample Line Group Properties (page 1149)

# Create Sample Lines - From Corridor Stations Dialog Box

Use this dialog box to create sample lines along an alignment for specified corridor stations.

**General**

Alignment

Displays the name of the currently selected alignment associated with the range of stations.

**Station Range**

From Alignment Start

Specifies whether the station range starts at the beginning of the alignment (that is, at the station located at the beginning of the alignment):

■ **True**: Begins the station range at the start of the alignment.

■ **False**: Specifies the beginning station in the range. Enter or pick a beginning station.

Start Station

Specifies the beginning station in the range. Enter a station value directly or click ⟊ to pick from the drawing. Available if From Alignment Start is False.

To Alignment End

Specifies whether the station range ends at the end of the alignment (that is, at the final station along the alignment):

■ **True**: Ends the station range at the end of the alignment.

■ **False**: Specifies the ending station in the range. Enter or pick an ending station.

End Station

Specifies the end station in the range. Enter a station value directly or click  to pick from the drawing. Available if To Alignment End is False.

### Left Swath Width

Snap To An Alignment

Specifies that the sample line snaps the swath widths to an offset alignment at each station:

■ **True**: Snaps the widths to an offset alignment at each station.

■ **False**: Does not snap the widths.

Alignment

Specifies the currently selected alignment. Available if Snap To Alignment is True.

**NOTE** Although all alignments in the current drawing are listed, do not select the parent alignment. The resulting length will be zero.

Width

Specifies the swath width to the left of the alignment, using the actual ground units. Enter a positive value in ground units. Available if Snap To Alignment is False.

### Right Swath Width

Snap To An Alignment

Specifies that the sample line snaps the swath widths to an offset alignment at each station:

■ **True**: Snaps the widths to an offset alignment at each station.

■ **False**: Does not snap the widths.

Alignment

Specifies the currently selected alignment. Available if Snap To Alignment is True.

**NOTE** Although all alignments in the current drawing are listed, do not select the parent alignment. The resulting length will be zero.

Width

Specifies the swath width to the right of the alignment, using the actual ground units. Enter a positive value in ground units. Available if Snap To Alignment is False.

**Related procedures:**

■ Creating Sample Lines (page 1142)

■ Editing Sample Line Group Properties (page 1149)

# Create Section View Wizard

Use this wizard to create section views.

**NOTE** Some controls on this wizard are not available when requisite objects are not present. For example, the Section Views Tables page of this wizard is not available if there is no materials list for the drawing. For information about how to create a material list, see Generating Material Lists (page 1163).

**Related procedures:**

■   Creating/Editing Section Views (page 1153)

## General Page (Create Section View Wizard)

Use this page to specify basic information about the section view, including the parent alignment and sample line group name, sample line, station, description, style, and layer.

Select Alignment

Specifies the name of the parent horizontal alignment. Select a name from the list or click  to pick an alignment from the drawing.

Sample Line Group Name

Specifies the parent sample line group of the sample line from which the section view is created. Select a name from the list or click  to pick an alignment from the drawing.

Sample Line

Specifies the sample line from which the section view is created. Select a name from the list or click  to pick an alignment from the drawing. Specifying a sample line also changes the Station value.

Station

Specifies the station value of the sample line from which the section view is created. Specifying a station also changes the Sample Line value.

Section View Name

Specifies an editable, system-generated name. The name must remain unique within the drawing.

Description

Specifies an optional description of the section view.

Section View Layer

Specifies the drawing layer for the section view. To review or change layer data, click  .

Section View Style

Specifies the style for the section view. Provides standard controls you use to review or change the style.

## Offset Range Page (Create Section View Wizard)

Use this page to specify the offset range to which the section view is drawn.

Automatic

Specifies that the offset range is set automatically. The lengths of the left and right swath widths are displayed in the fields next to the selection.

User Specified

Specifies that the offset range is set by the user. Specify the lengths of the left and right swath widths in the fields next to the selection.

Left

Specifies the offset length of the left swath width.

Right

Specifies the offset length of the right swath width.

## Elevation Range Page (Create Section View Wizard)

Use this tab to specify the elevation properties of the section view.

### Elevation Range

Automatic

Specifies that the elevation range is set automatically. The minimum and maximum heights are displayed in the fields next to the selection.

User Specified

Specifies that the elevation range is set by the user. Specify minimum and maximum heights in the fields next to the selection.

Minimum

Specifies the minimum height of the elevation.

Maximum

Specifies the maximum height of the elevation.

## Section Display Options Page (Create Section View Wizard)

Use this page to specify section display options.

### Select Sections To Draw

Name

Displays the name of the section.

Draw

Specifies whether the section is drawn in the section view. Select the check box to draw the section in the section view.

Clip Grid

Specifies whether the grid lines in the section view are clipped to the section (horizontal and/or vertical depending on the section view style definition). Click the radio button to clip the grid lines in the section view to one of the named sections.

Change Labels

Specifies the style set from which the section labels will be drawn. Click in this column to open the Select Style dialog box (page 1805)

Style

Specifies the section style. Click in this column to open the Select Section Style dialog box.

Override Style

Specifies the style for a section. Select the check box to open the Select Section Style dialog box.

## Data Bands Page (Create Section View Wizard)

Use this page to specify the properties of the data bands associated with the section view.

Select Band Set

Specifies the set of band styles for the section view. Provides standard controls that you use to review or change the set.

### List of Bands

Location

Specifies either the top or bottom of the section view. The table below this field shows the location of the bands in the band style set.

Band Type

Specifies the data band type, either Section Data or Section Segment.

Style

Specifies the style for the band type.

Surface 1

Specifies a surface sampled by the sample line that supplies the data for the band, including any corridor surface.

Surface 2

Specifies an additional surface sampled by the sample line that supplies the data for the band, including any corridor surface. Not used for Section Segment type data bands.

## Section View Tables Page (Create Section View Wizard)

Use this page to set volume table properties for the section view.

---

**NOTE** This page is not available if there is no material list for the drawing. For information about how to create a material list, see Generating Material Lists (page 1163).

---

Use the following controls to specify the volume table types to draw:

Type

Specifies the type of the volume table(s).

Select Table Style

Specifies the style of the volume table. Standard buttons are also provided to create or edit a label style.

Add >>

Adds a volume table to the list.

### List Of Volume Tables

Table Type

Displays the table type.

Style

Specifies the name of the table style that was added using the Select Table Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to either create a new style, copy or edit the style selection, or pick a style from the drawing.

Material List

Specifies the material list for the volume table.

Materials

Click ⋯ to specify the materials for the volume table.

Layer

Specifies the drawing layer that contains the sampled source.

Split

Specifies whether the layer is split.

Gap

Specifies the distance in plotted units between the data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

Reactivity Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically when either the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

**Position of Table(s) Relative to Section View**

Section View Anchor

Specifies the section view anchor (Top/Middle/Bottom, Left/Center/Right).

Table Anchor

Specifies the location of the table anchor (Top/Middle/Bottom, Left/Center/Right).

Table Layout

Specifies whether the layout of the tables will be horizontal or vertical, in relation to the section view.

X Offset

Specifies the distance in plotted units to move the title away from the position set by the table anchor. Enter a positive number to move the title right, or a negative number to move it left.

Y Offset

Specifies the distance in plotted units to move the title away from the position set by the table anchor. Enter a positive number to move the title up, or a negative number to move it down.

# Edit Feature Settings - Sample Line Dialog Box

Use this dialog box to view and change sample line-related settings.

This topic documents settings in all sample line-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the drawing icon.

- Sample line feature settings are listed near the top of this dialog box, after the General property group, and are identified by the sample line icon.

- Sample line command settings are identified by the command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

Use these settings to establish the default styles assigned to sample line components:

Sample Line Style

Specifies the default sample line style. Click in the Value column, and click ⊡ to select a style in the Sample Line Style dialog box.

Sample Line Label Style

Specifies the default sample line label style. Click in the Value column, and click ⊡ to select a style in the Sample Line Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### Default Name Format

Use these settings to specify the default name formats for new sample lines and sample line groups. Click in the Value column, and click ⊡ to make changes in the Name Template dialog box (page 1806).

### Default Swath Widths

**NOTE** This property group is displayed when you access the settings from the CreateSampleLine command.

Use these settings to specify the default behavior for sample line swath width.

Left Swath Width

Specifies the swath width to the left of the alignment, using the actual ground units. Enter a width in the Value column or click 🔲 and select a distance in the drawing area.

Right Swath Width

Specifies the swath width to the right of the alignment, using the actual ground units. Enter a width in the Value column or click 🔲 and select a distance in the drawing area.

### Sampling Increments

**NOTE** This property group is displayed when you access the settings from the CreateSampleLine command.

Use these settings to specify the default behavior for sampling increments.

**NOTE** Changing sampling increments can affect performance on rebuild. A shorter increment means more sampling stations, and thus a larger drawing.

Use Sampling Increments

Specifies whether sample lines are created at incremental stations along the alignment.

Increment Along Tangents

Specifies the sample increment distance along tangents, using the actual ground units. Enter a distance in the Value column or click 🔲 and select a distance in the drawing area.

Increment Along Curves

Specifies the sample increment distance along curves, using the actual ground units. Enter a distance in the Value column or click 🔲 and select a distance in the drawing area.

Increment Along Spirals

Specifies the sample increment distance along spirals, using the actual ground units. Enter a distance in the Value column or click 🔲 and select a distance in the drawing area.

### Additional Sample Controls

**NOTE** This property group is displayed when you access the settings from the CreateSampleLine command.

Use these settings to specify the default behavior for additional sample controls.

At Range Start

Specifies whether a sample line is created at the beginning of the specified range.

At Range End

Specifies whether a sample line is created at the end of the specified range.

At Horizontal Geometry Points

Specifies whether additional sample lines are created at horizontal geometry points, such as, at the start of a spiral, or at a spiral-curve point within the current range.

At Superelevation Critical Stations

Specifies whether additional sample lines are created at superelevation critical stations within the current range.

Start Range At Alignment Start

Specifies whether the sampling range starts at the beginning of the alignment.

End Range At Alignment End

Specifies whether the sampling range ends at the end of the alignment.

### Miscellaneous

**NOTE** This property group is displayed when you access the settings from the CreateSampleLine command.

Use this setting to specify miscellaneous default behavior for sample lines.

Lock To Station

Specifies whether the sample lines update when the alignment geometry and properties change.

**Related procedures:**

■  Sample Line Settings (page 1128)


# Edit Feature Settings - Section Dialog Box

Use this dialog box to view and change section-related settings.

This topic documents settings in all section-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■  Drawing-level ambient settings are identified by the ⬜ drawing icon.

■  Section feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ⬜ section icon.

■  Section command settings are identified by the ⬜ command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

Section Style

Specifies the default section object style. Click in the Value column, and click  to select a style in the Section Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use these settings to specify the default name formats for new section object. Click in the Value column, and click  to make changes in the Name Template dialog box (page 1806).

**Related procedures:**

■  Section Settings (page 1129)

# Edit Feature Settings - Section View Dialog Box

Use this dialog box to view and change section view-related settings.

This topic documents settings in all section view-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■  Drawing-level ambient settings are identified by the  drawing icon.

■  Section view feature settings are listed near the top of this dialog box, after the General property group, and are identified by the  section view icon.

■  Section view command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

Use these settings to specify the default styles assigned to section view components:

Marker Style

Specifies the default marker style for section view points for offset elevation and depth grade labels. Click in the Value column, and click  to select a style in the Marker Style dialog box.

Section View Offset Elevation Label Style

Specifies the default section view offset elevation label style. Click in the Value column, and click  to select a style in the Section View Offset Elevation Label Style dialog box.

Section View Depth Grade Label Style

Specifies the default section view grade label style. Click in the Value column, and click  to select a style in the Section View Depth Grade Label Style dialog box.

Section View Style

Specifies the default section view style. Click in the Value column, and click [...] to select a style in the Section View Style dialog box.

Section View Band Set

Specifies the default section view band set style. Click in the Value column, and click [...] to select a band set in the Section View Band Set dialog box.

Section Label Set

Specifies the default section label set style. Click in the Value column, and click [...] to select a label set in the Section Label Set dialog box.

Group Plot Style

Specifies the default group plot style for the multiple section view objects. Click in the Value column, and click [...] to select a style in the Group Plot Style dialog box.

Section Projection Label Style

Specifies the default style for labels on projected objects. Click in the Value column, and click [...] to select a style in the Section Projection Label Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use these settings to specify the template for naming the section view.

Section View Name Template

Specifies the default section view name template.
The default format is <[Section View Station (Uft|FS|P2|RN|Sn|OF|AP|B2|TP|EN|W0|DZY)]> (<[Next Counter(CP)]>).
The Section View Station name is followed by a number of parameters that define how the text is displayed:

- **U—Units**: Select one of ft (foot), m (meter), mile (mile), km (kilometer), in (inch), or yd (yard). The default is Uft.

- **F—Format**: Select either S (station format) or D (DD.DDDDDD decimal). The default is FS.

- **P—Precision**: Select one of 0 (1), 1 (0.1), 2 (0.01), ... , or 8 (0.00000001). The default is P2.

- **R—Rounding**: Select one of N (round normal), U (round up), or T (truncate). The default is RN.

- **S—Sign**: Select one of n (sign negative '-'), Bn ((Bracket negative)), a (sign always ( )), D (drop sign), BL ('(' left prentice), or BR (right prentice '('). The default is Sn.

- **O—Output**: Select one of F (full), LD (left of decimal), RD (right of decimal), DS (decimal character), LB (left of station character), RB (right of station character), BBD (between station character and decimal), or BC (station character). The default is OF.

- **A—Decimal character**: Select either P ('.' period) or C (',' comma). The default is AP.

- **B—Station character position**: Select one of 1 (1+0), 2 (1+00), 3 (1+000), 4 (1+0000), or 5 (1+00000). The default is B2.

- **T—Station character**: Select one of P (plus sign '+'), M (minus sign '-'), A (automatic ( )), U (underscore '_'), or N (none). The default is TP.

- **E—Drop decimal for whole number**: Select either N (No) or Y (Yes). The default is EN.

- **W—Minimum display width**: Select one of 0 (none), 1, 2, ..., 32. The default is W0.

- **DZ—Drop leading zeros**: Select one of Y (yes) or N (no). The default is DZN.

For example, a section view name, using the above defaults and following the "rules" of the parameters, could look like this:
2+50.00 (2)

Click in the Value column, and click ⬚ to make changes in the Name Template dialog box (page 1806).

## Section View Creation

Use these settings to establish the defaults assigned for section view creation:

Specify Section View Offset Range

Specifies whether an offset range is set for the creation of a section view.

Specify Section View Height

Specifies whether the default section view height is set for the creation of a section view.

Section Group Elevation Range

Specifies how the elevation range of the section group is set:

- Lowest Elevation

- Mean Elevation

- Follow a Section

## Default Projection Label Placement

Use these settings to specify the default placement of labels for objects projected to section views:

Dimension Anchor Option

Specifies whether the default anchor option will be Fixed, Above, Below, Graph View Top, or Graph View Bottom.

Dimension Anchor Elevation Value For Projections

Specifies the default value for anchor elevation. Enter a value or click 🔲 and select an elevation in the drawing area.

Dimension Anchor Plot Height Value For Projections

Specifies the default value for anchor plot height. Enter a value or retain the existing default value.

## Table Creation

**NOTE** This property group is displayed when you access the settings from the CreateMultipleSectionView or CreateSectionView commands.

Use these settings to specify defaults for creating tables with section views.

Split Table

Specifies whether a table is split into two or more sections after a specified maximum number of rows has been met.

Maximum Rows Per Table

Specifies the maximum number of rows to include per section. If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of sections to include in each stack.

Table Spacing
   Specifies the spacing between tables.

Total Volume Table Style

   Specifies the style for a total volume table. Click in the Value column, and click ⊡ to select a style in the
   Total Volume Table Style dialog box.

Material Table Style

   Specifies the styles for a material table. Click in the Value column, and click ⊡ to select a style in the
   Material Table Style dialog box.

X Offset
   Specifies the horizontal offset of the table.

Y Offset
   Specifies the vertical offset of the table.

Tile Direction
   Specifies the direction in which the table tiles (across or down).

Section View Anchor
   Specifies the section view anchor (Top/Middle/Bottom, Left/Center/Right).

Table Anchor
   Specifies the location of the table anchor (Top/Middle/Bottom, Left/Center/Right).

Table Layout
   Specifies whether the layout of the tables will be horizontal or vertical, in relation to the section view.

## Multiple Section View Creation

**NOTE** This property group is displayed when you access the settings from the CreateMultipleSectionView command.

Use these settings to specify defaults for creating multiple section views.

Specify Multiple Section View Station Range
   Specifies whether a station range is set for the creation of multiple section views.

Default Section VIew Height

   Specifies the default height for section views. Enter a height in the Value column or click ⬛ and select
   a height in the drawing area.

**Related procedures:**

- Section View Settings (page 1130)


# Edit Offset and Elevation Ranges Dialog Box

Use this dialog box to specify the offset and elevation range properties for selected section view groups.

**Related procedures:**

- Editing Section View Properties (page 1156)

# Offset Range Tab (Edit Offset and Elevation Ranges Dialog Box)

Use this tab to specify the offset range to which the section views are drawn.

Automatic

Specifies that the offset range is set automatically. The lengths of the left and right swath widths are displayed in the fields next to the selection.

User Specified

Specifies that the offset range is set by the user. Specify the length of the left and right swath widths in the fields next to the selection.

Left

Specifies the offset length of the left swath width.

Right

Specifies the offset length of the right swath width.

# Elevation Range Tab (Edit Offset and Elevation Ranges Dialog Box)

Use this tab to specify the elevation properties of the selected section view.

## Elevation Range

Automatic

Specifies that the elevation range is set automatically. The minimum and maximum heights are displayed in the fields next to the selection.

User Specified

Specifies that the elevation range is set by the user. Specify how the elevation is applied by selecting a Section Views Height Option.

Height

Specifies the height of the elevation.

## Section Views Height Option

From Lowest Elevations Of All Sections

Specifies that the elevation is set from the lowest elevation of each section.

From Mean Elevations Of All Sections

Specifies that the elevation is set from the mean elevation of each section.

Follow A Section

Specifies that the elevation is set from a particular section. Select the section from the Select Section list.

# Edit Sample Line Dialog Box

Use this dialog box to edit some of the values for the parameters of each of the vertices on an existing sample line.

**NOTE** This dialog box is populated with parameters and values only when a valid sample line is selected.

## Information

Name

Specifies the name of the selected sample line.

**General**

Sample Line Vertex Name

Displays the sample line vertex name.
For example, in a straight orthogonal sample line with positive left and right swath width values across an alignment, the vertex names would be Left 1, Center, and Right 1.

Sample Line Vertex Northing

Displays the sample line vertex northing value.

Sample Line Vertex Easting

Displays the sample line vertex easting value.

**Default Swath Widths**

Left Swath Width

Specifies the width of the left swath

Right Swath Width

Specifies the width of the right swath

**Station**

Sample Line Station Value

Specifies the station value of the selected sample line.

**Navigation**

Previous Vertex

Click to display the previous vertex. The previous vertex parameters and values for the selected sample line are displayed.

Next Vertex

Click to display the next vertex. The next vertex parameters and values for the selected sample line are displayed.

**Related procedures:**

■ Editing Sample Lines (page 1145)

# Group Plot Style Dialog Box

Use this dialog box to edit settings that control the layout of multiple section views.

**See also:**

■ Creating and Editing Section View Styles (page 1133)

# Information Tab (Group Plot Style Dialog Box)

Use this tab to display administrative data and edit primary information about the group plot style, such as its name and description.

This tab also contains information about who created or modified the group plot style and the date of these events. The following fields can be modified:

Name

Specifies the name of the selected sheet style.

Description
   Specifies the optional description of the selected sheet style.

# Array Tab (Group Plot Style Dialog Box)

Use this tab to edit settings for the plot rules and page layout features for the section views.

## Plot Rules

Specifies whether to plot the section views by rows or columns.

By Rows
   Plots section views in rows as specified.

By Columns
   Plots section views in columns as specified.

Start Corner
   Specifies the starting corner for plotting the section views.

Align Section Views About
   Specifies how the section views are aligned. Select one:

   ■ Left

   ■ Centerline

   ■ Right

Cell Sizes
   Specifies how the size of each cell in the array will be determined. For this item, the term "cell" refers to the rectangular area in an array of section views in which a single section view is displayed. Select one:

   ■ Uniform For All: When this option is selected, each section view in the array will be displayed in an area (cell) that is the same size for all section views in the array. The size of each cell will be the size of the tallest and the widest section view in the array. Therefore, when this option is selected, all cells in the array will be the same size.

   ■ Uniform Per Row Or Column: When By Rows is selected at the top of this dialog box, and you also select this option, each row in the array will be fit to the height of the tallest section view in the row. Therefore, when this option is selected, each row in an array may have a different height. However, within each row, all section views will display in an area with the same cell height (row height). When By Column is selected at the top of this dialog box, and you also select this option, each column in the array will be fit to the width of the widest section view in the column. Therefore, when this option is selected, each column in an array may have a different width. However, within each column, all section views will display in an area with the same cell width (column width).

## Space Between Adjacent Section Views

Column
   Specifies the spacing between columns.

Row
   Specifies the spacing between rows.

# Plot Area Tab (Group Plot Style Dialog Box)

Use the settings on this tab to specify the method of plotting section views (either all or by page), the sheet style to use, and the gap between successive pages.

Plot All
>Plots all of the section views on a single page.

Plot By Page
>Plots the section views using the specified sheet style and gap values.

Sheet Style
>Specifies the sheet style. Select a sheet style in the list.

Gap Between Successive Pages
>Specifies the plotted distance between each successive page in a multiple section view plot. Enter a positive value in plotted units.

## Summary Tab (Group Plot Style Dialog Box)

Use this tab to edit settings for the group plot style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Pipe Network Section Properties Dialog Box

Use this dialog box to edit properties of the selected pipe network section, such as its name, section data, and labels.

**See also:**

- Editing Pipe Network Section Properties (page 1152)
- Displaying Pipe Networks in Section Views (page 1221)

## Information Tab (Pipe Network Section Properties Dialog Box)

Use this tab to edit primary information about the selected pipe network section.

Name
>Specifies the name of the section. The default format is <SLG>-n - <SL>-n - <data source>(n).
><SLG> is the parent sample line group name, <SL> is the sample line name, and <data source> is the name of the data source (for example, pipe network) on which the section is drawn.
>n is an integer greater than or equal to one, which increments as each feature is created in the drawing.

Description
>Specifies the optional description of the section.

Show Tooltips
>Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Section Data Tab (Pipe Network Section Properties Dialog Box)

Use this tab to edit properties of the selected section object.

Description
>Specifies the optional description of the current section.

Type
>Displays a , which indicates that the section was extracted from a pipe network.

Data Source

Displays the name of the data source from which the current section was sampled.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface or corridor) geometry changes.

Layer

Specifies the drawing layer that contains the current section associated with the selected sample line group. Click in this column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the name of the style of the section. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Station

Displays the station value of the sample line along which the section is sampled.

Left Offset

Displays the left offset value of the current section.

Right Offset

Displays the right offset value of the current section.

Minimum Elevation

Displays the minimum elevation of the section associated with the selected sample line group.

Maximum Elevation

Displays the maximum elevation of the section associated with the selected sample line group.

# Profile Grade Points Dialog Box

Use this dialog box to select a profile and display its elevation by marking it at the appropriate offset and elevation in the section view.

Alignment

Specifies the parent alignment whose profile grade point is to be displayed in this section view. Select an alignment from the drop-down list or click  to pick an alignment from the drawing.

Add >>

Adds the profile grade point for the currently selected alignment to the list.



Deletes the highlighted profile grade point from the list. The profile grade point will no longer be displayed in the section view.

**List of profile grade points**

Alignment

Displays the name of the parent alignment whose profile grade point is marked in this section view.

Show

Specifies whether the marker is visible in the section view (that is, without having to edit the marker style or layer).

Profile

Displays the name of the profile whose location, with respect to the current section view, is marked.

Marker Style

Displays the marker style used for the profile grade point.

**Related procedures:**

■ Creating/Editing Section Views (page 1153)

## Project Objects to Section View Dialog Box

Use this dialog box to specify objects in the current drawing to be projected into a section view.

To highlight an object in the drawing, select its row in this table.

Name

Displays the objects that can be projected, divided into categories. Select the ones you want to project.

Style

Specifies the current style for each object. You can change the style used to project an object. Select <set all> at the category level to change all objects in that category in one operation.

Elevation Options

Specifies how the elevation of each object is determined.

■ Use Object: Elevation is read from the object properties.

■ Surface: The object is set to the elevation of a selected surface.

■ Manual: You can specify the elevation. This option cannot be used with longitudinal objects.

■ Section: The object is set to the elevation of a selected section.

■ <set all>: Used at the category level to specify the elevation option for all projected objects in the category.

Elevation Value

Specifies the elevation of each projected object. You can change the value here for point objects if the elevation option is manual.

Label Style

Specifies the style of label applied to the projected object. Select <none> to display no label. Select <set all> at the category level to set a label style for all projected objects in the category.

 Pick Objects

Click to select objects in the drawing and add them to the section view.

**Related procedures:**

■ Working with Projected Objects (page 1118)

## Resolve Duplicate Sample Lines Dialog Box

Use this dialog box to resolve duplicate sample lines that have already been defined at some stations.

The properties in this dialog box specify whether duplicate sample lines are deleted from an existing list or appended to a new list. If you select Delete Existing List, you need not do anything further. If you select Append New List, you can specify how the new list is appended.

## Delete Existing List

Removes the existing sample lines in the drawing. Adds only new sample lines.

## Append New List

Station

Displays the station where the duplicate sample line is located. This is the default.

Overwrite

Specifies whether the sample line to be created overwrites the existing sample line in the drawing:

- Selected: The sample line to be created overwrites the existing sample line in the drawing, thus removing the existing sample line at that station.

- Cleared: The sample line to be created does not overwrite the existing sample line, thus preserving the existing sample line.

Add New

Specifies whether the sample line is added as a new sample line in the drawing:

- Selected: The sample line to be created is added as a new sample line in the drawing, thus keeping the existing sample line.

- Cleared: The sample line to be created is not added as a new sample line, thus keeping the existing sample line.

Ignore New

Specifies whether the sample line to be created is ignored and not added in the drawing:

- Selected: The sample line to be created is ignored and not added, thus preserving only the existing sample line in the drawing.

- Cleared: The sample line to be created is not ignored and is added, thus preserving the existing sample line in the drawing.

Name

Specifies the name of the sample line.

Description

Specifies the description of the sample line.

Overwrite All

Specifies that all duplicate sample lines in the list are overwritten in the drawing. Optionally select the Overwrite check box for all duplicate sample lines. This is the default.

Add All

Specifies that all duplicate sample lines in the list are added in the drawing. Optionally select Add New check box for all duplicate sample lines.

Ignore All

Specifies that all duplicate sample lines in the list are ignored, and not added in the drawing. Optionally select the Ignore All check box for all duplicate sample lines.

# Sample Line Group Properties Dialog Box

Use this dialog box to edit properties of the selected sample line group, such as its name, and properties of the displayed sections and section views.

**See also:**

- Editing Sample Line Group Properties (page 1149)

## Information Tab (Sample Line Group Properties Dialog Box)

Use this tab to edit primary information about the selected sample line group.

Name

Specifies the name of the selected sample line group. The default format is <SLG>-n.
<SLG> is the parent sample line group name.
n is an integer greater than or equal to one, which increments as each feature is created in the drawing.

Description

Specifies an optional description.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Sample Lines Tab (Sample Line Group Properties Dialog Box)

Use this tab to edit properties of the sample lines that are included in the selected sample line group.

Edit Group Labels

Click to open the Sample Line Labels dialog box (page 2261).

**Sample Lines**

No.

Displays the sequential number of the sample line.

Name

Displays the name of the sample line.

Station

Displays the station that is associated with the sample.

Layer

Specifies the layer on which the sample line is drawn. The default is 0 (base), which is the default layer defined in the drawing layer settings. Click in the Layer column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the sample line style. Click in the Style column to open the Pick Sample Line Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Left Offset

Displays the left offset value of the current selection.

Right Offset

Displays the right offset value of the current selection.

Description

Displays the description of the sample line.

## Sections Tab (Sample Line Group Properties Dialog Box)

Use this tab to edit properties of sections that are included in the selected sample line group.

Sample More Sources

Click the Sample More Sources button to open the Section Sources dialog box (page 2273).

### Sections List

Name

Specifies the name of the section associated with the selected sample line group.

Style

Specifies the name of the style of the section. Click in the column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

Layer

Specifies the drawing layer that contains the section associated with the selected sample line group. The default is 0 (base), which is the default layer defined in the drawing layer settings. Click in the Layer column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Station

Displays the station value of the sample line along which the current section is sampled.

Show Sections For All Sources

Select the Show Sections for all Sources check box to view all stations of each section, arranged in an expandable tree or hierarchy.

### Related procedures:

■ Setting Up Sections

## Section Views Tab (Sample Line Group Properties Dialog Box)

Use this tab to edit properties of the section views in the selected sample line group.

Section View

Displays the names of section views arranged in an expandable tree or hierarchy.

Group Plot Style

Specifies the name of the plot style of the section view associated with the selected sample line group. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Style

Specifies the name of the style of the section view associated with the selected sample line group. Click in this column to open the Select Style dialog box. Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Change Band Set

Specifies the optional set of bands to use with the current section view. Click in this column to open the Select Style dialog box. Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Change Volume Tables

Specifies the volume table(s) to be drawn. Click in this column to open the Change Volume Table dialog box (page 2299). A material list must be exist for the drawing before any volume table can be created.

Section Display

Specifies the section display options for the current section view. Click in this column to open the Section Display Controls dialog box (page 2269). Select section display options in the list.

Profile Grade

Specifies the grid lines displayed at offset alignments and other profiles within the range of section view extents. Click in this column to open the Profile Grade Points dialog box (page 2254).

Station

Displays the station value of the sample line along which this section view is created.

Start Station

Displays the beginning station in the range.

End Station

Displays the end station in the range.

# Material List Tab (Sample Line Group Properties Dialog Box)

Use this tab to view and edit the material lists for a sample line group and add and delete materials from a list.

Add New Material

Adds a new empty criteria to a material list. After you add the criteria, you can edit its quantity type and settings and use the Define Material fields to populate it with data.



Opens the Name Template dialog box (page 1806), where you can modify the material naming template.



Deletes the material or data component that is currently selected in the Material Name column.

**Define Material**

Data Type

Specifies the type of data that is compared and processed when defining the material. Either Surface or Corridor Shape.

Select Surface/Shape

Lists sampled surfaces and corridor shapes for this sample line group. Select a surface or corridor shape.

**NOTE** Corridor shapes can be added only to a material with a Structure quantity type.

Adds the data specified in the Define Material fields to the selected material type.

**NOTE** If a material is not selected, the data is not added.

The properties table contains the following columns:

Material Name

Names of material lists, material names, surfaces, and corridor shapes arranged in a tree or hierarchy. Each instance of applying criteria to materials in a sample line group is added as a numbered list. List names can be edited.

Click ⊞ next to a list name to display its components (materials). Click ⊞ next to a material name to display its components (surfaces or structures).

Condition

Specifies the condition on which to base the calculation:

■ **Above**. Specifies that an area above this surface is included in the material definition. Used with Below to define two or more surfaces for cut, fill, and structures material types.

■ **Below**. Specifies that an area below this surface is included in the material definition. Used with Above to define two or more surfaces for cut, fill, and structures material types.

■ **Base**. Specifies that this surface is the surface to compare against the compare surface. Used with Compare to define two or more surfaces for earthworks and cut and fill material types.

■ **Compare**. Specifies that this surface is the surface to compare against the base surface. Used with Base to define two or more surfaces for earthworks and cut and fill material types.

■ **Include**. Specifies a corridor shape that is included in the structure type definition.

Quantity Type

Specifies the quantity type:

■ **Cut**. Calculates the material to remove.

■ **Fill**. Calculates the material to add.

■ **Cut and Refill**. Defines an area in the section where a material is removed and refilled with fill material.

■ **Earthworks**. Compares two surfaces to calculate both cut and fill areas and displays them separately.

■ **Structures**. Calculates the volume of one or more corridor shapes (as defined by the shape codes that are used to define the corridor). For information about shape codes, see Understanding Point, Link, and Shape Codes (page 1406).

Cut Factor

Specifies the expansion or swell of the cut material.

Fill Factor

Specifies the contraction or shrinkage of the fill material.

Refill Factor

Specifies usability factor used to calculate how much cut material can be reused as fill.

Shape Style

Specifies the default style used to display the material in a section view.

Curve Tolerance

Specifies the curve correction tolerance, if any, that was specified when the criteria was applied. To edit the value, select the check box, and then edit the value. Used only for List entries.

Import Another Criteria

Prompts you to add a new materials list to the sample line group. Prompts you to select a criteria and opens the Compute Materials dialog box.

# Sample Line Labels Dialog Box

Use this dialog box to manage labels for a particular sample line group.

Sample Line Label Style

Specifies the style for the labels. Standard buttons are also provided to create or edit a label style.

Add>>

Adds the specified label type to the set in the label table. These labels appear along the section line in the section view.

✖

Deletes the selected label type from the table. Also removes it from the section.

**Label table**

Type

Specifies the type of section label.

Style

Specifies the label style.

Start Station

Specifies the start station of the sample line group to be labeled.
The check box in front of the column specifies whether the start station can be edited:

■ **Selected**: The start station entry is frozen.

■ **Cleared**: The start station entry can be edited.

End Station

Specifies the end station of the sample line group to be labeled.
The check box in front of the column specifies whether the end station can be edited:

■ **Selected**: The end station entry is frozen.

■ **Cleared**: The end station entry can be edited.

**Related procedures:**

■ Using Sample Line Labels (page 1136)

# Sample Line Properties Dialog Box

Use this dialog box to edit properties of the selected sample line, such as its name, sample line data, and properties of the displayed sections and section views.

**See also:**

■ Editing Sample Line Properties (page 1149)

# Information Tab (Sample Line Properties Dialog Box)

Use this tab to edit primary information about the selected sample line.

Name

Specifies the name of the sample line. The default format is <SL>-n.
<SL> is the name of the sample line.
n is an integer greater than or equal to one, which increments as sample lines are created in the drawing.

Description

Specifies the optional description of the sample line.

Object Style

Specifies the name of the style used by the sample line.



Creates, copies, or edits a style using the Sample Line Style dialog box (page 2264). Click the Down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.

■ **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



Displays details about an existing style. Select the style name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

# Sample Line Data Tab (Sample Line Properties Dialog Box)

Use this tab to display properties for the sample line and to edit properties about the lock method and label style.

Group Name

Displays the name of the parent sample line group for the current sample line.

Alignment

Displays the name of the parent horizontal alignment for the current sample line.

Sample Line Number

Displays the assigned sample line number of the currently selected sample line.

**NOTE** This is a sequential number of the sample line among all sample lines sorted by station value.

Station

Displays the station at which the sample line intersects with the parent alignment.

Lock To The Station

Locks the sample line to the station:

- **Selected**: If the parent alignment geometry is updated so that the current station value is at a different XY, then the sample line moves to that location. The sample line maintains its relative position to the alignment (that is, it "slides" to the new location.

- **Cleared**: If the parent alignment is modified, the sample line remains in its location. The sample line maintains its geometry and recomputes the updated station where it intersects the alignment. Subsequently, if the check box is (re-)selected, the sample line is locked to the current station.

## Sections Tab (Sample Line Properties Dialog Box)

Use this tab to edit properties that draw and manage the section associated with the current sample line.

Name

Displays the name of the section associated with the current sample line.

Description

Specifies the optional description of the section.

Type

Displays the type of data source from which the section data was extracted. Either , which indicates from an existing TIN surface; , which indicates from a corridor; , which indicates from a corridor surface; or , which indicates a pipe network.

Data Source

Displays the name of the data source used by the section associated with the selected sample line. Can be a surface, corridor, corridor surface, or pipe network.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

Layer

Specifies the drawing layer that contains the section object associated with the selected sample line. Click in this column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the name of the style of the section. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Left Offset

Displays the left offset value of the current section.

Right Offset

Displays the right offset value of the current section.

Minimum Elevation

Displays the minimum elevation of the section associated with the selected sample line group.

Maximum Elevation

Displays the maximum elevation of the section associated with the selected sample line group.

Station

> Displays the station value at which the sample line is created.

## Section Views Tab (Sample Line Properties Dialog Box)

Use this tab to edit properties of the section views that are associated with the sample line.

Name

> Displays the name of the section view that is created along this sample line.

Description

> Displays the optional description of the section view.

Layer

> Specifies the drawing layer that contains the section view object. Click in this column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

> Specifies the name of the style used by the section view. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Change Band Set

> Specifies the optional set of bands to use with the current section view. Click in this column to open the Select Style dialog box. Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Sample Line Number

> Displays the current sample line number.

Sample Line Name

> Displays the current sample line name.

Station

> Displays the current sample line station value.

# Sample Line Style Dialog Box

Use this dialog box to define a sample line style, which controls the way a sample line is displayed in a drawing.

**See also:**

■ Creating and Editing Sample Line Styles (page 1131)

## Information Tab (Sample Line Style Dialog Box)

Use this tab to display administrative data and edit primary information about the sample line style, such as its name and description.

This tab also contains information about who created or modified the sample line style and the date of these events. The following fields can be modified:

Name

> Specifies the name of the selected sample line style.

Description

Specifies the optional description of the selected sample line style.

## Display Tab (Sample Line Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of sample line components, such as lines and vertices.

On the Display tab, you can use the ByLayer or ByBlock reference settings to set Color, Linetype, Lineweight, and Plot Style values. Reference settings are indirect and require some planning.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Sample Line Style Dialog Box)

Use this tab to edit settings for the sample line style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

## Sample Line Tools Toolbar

Use this toolbar to create and edit sample lines along a horizontal alignment.

The current sample line creation method, the alignment name associated with the sample line, and the layer on which the sample line is contained are displayed at the bottom of the tool.



Name

Specifies the format of the name of the next sample line to be created. The default format is <SL> - <[Next Counter]>.
<SL> is the name of the sample line.
<[Next Counter]> is an integer greater than or equal to one, which increments as sample lines are created in the drawing. This naming convention is based on the name template.
Enter a name or edit a default naming convention in the Name Template dialog box. For more information, see Name Template dialog box (page 1806).



Opens the Name Template dialog box. Edit the default sample line naming convention.



Prompts you to pick the horizontal alignment in the drawing to associate with the sample line. After you click the alignment in the drawing, the Sample Line Tools toolbar is reopened.

Current Sample Line Group

Displays the current sample line group with which the sample line is associated. If no sample line groups have been defined, you are prompted to enter a new sample line group name. For more information, see the Create/Edit Sample Line Group dialog box (page 2231).

**TIP** To edit a specific sample line group, select the sample line group name in the list.

Sample Line Group Actions

Specifies the sample line group action used to create or modify the sample line group. The default is the last used method. Click the down arrow to display other actions.

- ■ Opens the Create Sample Line Group dialog box. Create a new sample line group.

- ■ Opens the Edit Sample Line Group dialog box. Edit the currently selected sample line group.

- ■ Deletes the currently selected sample line group.

- ■ Prompts you to pick a sample line group from the drawing, which becomes the current sample line group.

- ■ Opens the Edit Sample Line Widths dialog box. Edit the left and right swath widths of the currently selected sample line group. The swath widths of individual sample lines are extended or trimmed to match the group's widths.

- ■ Opens the Section Sources dialog box. Add more source sections to the currently selected sample line group.

Sample Line Creation Methods

Displays the various sample line creation methods used to create the sample line(s). Click the down arrow to display other actions.

- ■ Creates sample lines by specifying individual stations along the alignment.

- ■ Creates sample lines by picking points in the drawing.

- ■ Creates sample lines by picking existing polylines in the drawing.

- ■ Opens the Create Sample Lines - By Station Range dialog box (page 2236). Creates sample lines by specifying a range of stations.

- ■ Opens the Create Sample Lines - From Corridor Stations dialog box (page 2238). Creates sample lines from corridor stations.

Click to select a sample line and view and edit its information in the Edit Sample Line dialog box (page 2250).

Toggles the display of the Edit Sample Line dialog box. To populate the columns of the Edit Sample Line dialog box, select a sample line from the drawing using .

Undoes the last action.

⇨

Redoes the last action. Limited to one operation.

**Related procedures:**

■ Creating Sample Lines (page 1142)

■ Editing Sample Lines (page 1145)

# Section Data Band Style Dialog Box

Use this dialog box to edit settings for section view band styles.

**See also:**

■ Section View Bands (page 1140)

■ Creating and Editing Section View Styles (page 1133)

## Information Tab (Section Data Band Style Dialog Box)

Use this tab to display administrative data and edit primary information about the section view band style, such as its name and description.

This tab also contains information about who created or modified the section view band style and the date of these events. The following fields can be modified:

Name

Specifies the name of the section data band.

Description

Specifies the optional description of the section view band style.

## Band Details Tab (Section Data Band Style Dialog Box)

Use this tab to specify the details of the section view band, including title text, layout, and labels and ticks values.

**NOTE** Both Section Data and Section Segment band styles are explained in this topic.

### Title Text

Compose Label

Click to create a title label in the Label Style Composer dialog box (page 1942).

### Layout

Band Height

Specifies the height of the band in plotted units. Enter a positive value in plotted units.

Text Box Width

Specifies the width of the band text box in plotted units. Enter a positive value in plotted units.

Offset From Band

Specifies the distance from the band to the text box in plotted units. Enter a positive value in plotted units.

Text Box Position
Specifies the location of the title text box with respect to the band.

## Labels And Ticks

Type
Specifies the type of labels and ticks.

---

**NOTE** To compose labels, select the type from the list. Adjust parameters as required. Click Compose Label.

---

The following types are available for section data bands:

- **At Major Increment**: Composes labels at major increments along the band and displays selected tick mark(s) at each major increment. The default is Small Ticks At, with Top and Bottom selected, each with a Tick Size of 0.1000".

- **At Minor Increment**: Composes labels at minor increments along the band and displays selected tick mark(s) at each minor increment. The default is Small Ticks At, with Top and Bottom selected, each with a Tick Size of 0.1000".

- **At Centerline**: Composes labels at centerline and displays selected tick mark(s) at centerline. The default is Full Band Height Ticks.

- **At Sample Line Vertices**: Composes labels at each sample line vertex along the band and displays selected tick mark(s) at each sample line vertex. The default is Small Ticks At, with Top and Bottom selected, each with a Tick Size of 0.1000".

- **At Grade Breaks**: Composes labels at each grade break along the band and displays selected tick mark(s) at each grade break. The default is Small Ticks At, with Top and Bottom selected, each with a Tick Size of 0.1000".

- **At Incremental DIstance**: Composes labels at incremental distances along the band. Click Compose Label to set label styles in the Label Style Composer dialog box (page 1942).

The following type is available for section segment bands:

- **Segment Label**: Composes labels at each segment along the band and displays selected tick mark(s) at each major increment. The default is Full Band Height Ticks.

When you select a Label and Tick type, the appropriate default values for Full Band Height Ticks and Small Ticks At are displayed.

Full Band Height Ticks
Specifies whether the tick is drawn at the full band height. This is the default value for the Centerline type.

Small Ticks At
Specifies whether ticks that are smaller than the size defined in Tick Size are drawn. The default for all types (except Centerline) is selected for both Top and Bottom and cleared for Middle. Select or clear one or more check boxes as required.

- **Top**: Draws the small tick(s) at the top of the selected type.

- **Middle**: Draws the small tick(s) in the middle of the selected type.

- **Bottom**: Draws the small tick(s) at the bottom of the selected type.

**Tick Size**: Specifies the height in plotted units of the Small Ticks. The default is 0.1000" for Top and Bottom ticks and 0.2500" for Middle ticks. Enter a positive value in plotted units.

Compose Label

Composes labels at major/minor increments, centerline, sample line vertices, and grade breaks for the section data band or section segment label. For more information, see Label Style Composer dialog box (page 1942).

## Display Tab (Section Data Band Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of section view components, such as borders, titles, ticks, and labels.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Section Data Band Style Dialog Box)

Use this tab to edit settings for the section view band style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Section Display Controls Dialog Box

Use this dialog box to edit section display options.

### Edit Section Options Table

Name

Displays the name of the section.

Draw

Specifies whether the section is drawn in the current section view. Select the check box to draw the section in the current section view.

Clip Grid

Specifies whether the grid lines in the current section view are clipped to the section (horizontal and/or vertical depending on the section view style definition). Select the radio button to clip the grid lines in the section view to the section.

Change Labels

Specifies the style set from which the section labels will be drawn. Click in this column to open the Select Style dialog box (page 1805).

Style

Specifies the section style. Click in this column to open the Select Section Style dialog box.

Override Style

Specifies the style for a section. Select the check box to open the Select Section Style dialog box.

# Section Editor Dialog Box

Use this dialog box to edit static sections in a section view and to display information about dynamic sections and corridor sections.

Static sections are those which do not maintain a link with their elevation data source.

This dialog box has a customizable display for the columns. For more information, see Customize Columns Dialog Box (page 1806).

Alignment
   Displays the alignment name associated with the current section.

Station
   Displays the station name associated with the current section.

**Columns**

No.
   Sequential number of the vertex.

Distance from CL
   Distance in ground units from the vertex to the centerline.

Vertex Elevation
   Elevation at the vertex.

Section Grade In
   Angle of the section grade in.

Section Grade Out
   Angle of the section grade out.

---

**NOTE** If you want to edit a dynamic section or a corridor section, you can make a copy of the section and change the type to static.

---

# Section Labels Dialog Box

Use this dialog box to manage labels for a particular section in a section view.

A set of labels created here can be saved and used for other section.

Type
   Specifies the type of label to be added to the section in the section view. One of:

   ■ **Major Offset**. Offsets, elevations, and instantaneous grades at major increments.

   ■ **Minor Offset**. Offsets, elevations, and instantaneous grades at minor increments.

   ■ **Segments**. Section segment labels for each (line) segment of a section object.

   ■ **Grade Breaks**. Grade break labels applied at every grade break point for a section.

Section <Label Type> Label Style
   Specifies the style for the labels. Standard buttons are also provided to create or edit a label style.

Add>>
   Adds the specified label type to the set in the label table. These labels appear along the section line in the section view.

✖
   Deletes the selected label type from the table. Also removes it from the section.

**Label table**

Type
   Specifies the type of section label.

Style
   Specifies the label style.

**Increment**

Specifies the distance between labels of this type. This is used only for station labels.

**Check box**

Indicates whether the labels are applied from the beginning of the section or start at a specified station:

- Selected: The label type is applied from the beginning of the section offset.

- Cleared: The label type is applied to the part of the section that begins at the offset recorded in the Start Offset column.

**Start Offset**

Specifies the offset where the label type is applied.

**Check box**

Indicates whether the labels are applied to the end of the section offset or finish at a specified offset.

- **Selected**: The label type is applied as far as the end of the section offset.

- **Cleared**: The label type is applied to a part of the section that ends at the offset recorded in the End Offset column.

**End Offset**

Specifies the offset where the label type is applied.

**Dim Anchor Opt**

Specifies the location of the anchor used to position dimension lines for the section label.

- **Distance Above**: Moves the dimension line anchor up the distance specified in the Dim Anchor Val field.

- **Fixed Elevation**: Places the dimension line anchor at a fixed elevation on a section view. Specify the elevation in the Dim Anchor Val field.

- **Distance Below**: Moves the dimension line anchor down the distance specified in the Dim Anchor Val field.

- **Graph View Top**: Places the line anchor at the top of the section view grid. This creates a vertical graph line above the label. To ensure the full line extent is displayed, set Dim Anchor Val to 0.

- **Graph View Bottom**: Anchors the line to the bottom of the section view grid. This creates a vertical graph line below the label. To ensure the full line extent is displayed, set Dim Anchor Val to 0.

**Dim Anchor Val**

Specifies the dimension line anchor value to be applied using the Dim Anchor Opt.

**Weeding**

Specifies a label exclusion distance for grade break labels only. Use this setting to remove overlapping labels, making it easier to read the remaining labels. If another grade break occurs within the specified distance, it is not labeled.

**NOTE** This setting is available for Grade Break labels only.

**Import Label Set**

Opens the Select Style dialog box (page 1805). Select an existing label set to use or to use as a basis for modification. This imported set overwrites the existing contents of the label table.

**Save Label Set**

Opens the Label Set dialog box (page 1771). Configure and save the current label set for use with other sections.

■  Using Section Labels (page 1137)

# Section Properties Dialog Box

Use this dialog box to edit properties of the selected section, such as its name, section data, and labels.

**See also:**

■  Editing Sections (page 1148)

■  Editing Section Properties (page 1150)

## Information Tab (Section Properties Dialog Box)

Use this tab to edit primary information about the selected section.

Name

Specifies the name of the section. The default format is <SLG>-n - <SL>-n - <data source>(n).
<SLG> is the parent sample line group name, <SL> is the sample line name, and <data source> is the name of the data source (for example, surface or corridor surface) on which the section is drawn.
n is an integer greater than or equal to one, which increments as each feature is created in the drawing.

Description

Specifies the optional description of the section.

Object Style

Specifies the name of the style used by the section.



Creates, copies, or edits a style using the Section Style dialog box (page 2274). Click the down arrow to display the style selection menu:

■  **Create New**: Creates a new style.

■  **Copy Current Selection**: Copies the current style.

■  **Edit Current Selection**: Edits the current style.

■  **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Section Data Tab (Section Properties Dialog Box)

Use this tab to edit properties of the selected section object.

Description

Specifies the optional description of the current section.

Type

Displays the type of data source from which the section data was extracted. Either ⌂ , which indicates from an existing TIN surface; ⌂ , which indicates from a corridor; ⌂ , which indicates from a corridor surface; or ⌂ , which indicates from an existing pipe network.

Data Source

Displays the name of the data source from which the current section was sampled.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source geometry changes.

Layer

Specifies the drawing layer that contains the current section associated with the selected sample line group. Click in the Layer column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the name of the style of the section. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Left Offset

Displays the left offset value of the current section.

Right Offset

Displays the right offset value of the current section.

Minimum Elevation

Displays the minimum elevation of the section associated with the selected sample line group.

Maximum Elevation

Displays the maximum elevation of the section associated with the selected sample line group.

Station

Displays the station value of the sample line along which the section is sampled.

# Section Sources Dialog Box

Use this dialog box to resample sections of a sample line group.

Sample Line Group Name

Displays the name of the sample line group being resampled.

Alignment Name

Displays the name of the parent alignment of the sample line group.

Add >>

Adds the specified sample line group to the Sampled Sources table.

Remove <<

Removes the specified sample line group from the Sampled Sources table.

**Sampled Sources Table**

Name
  Specifies the name of the sampled source.

Style
  Specifies the style of the sampled source.

Layer
  Specifies the drawing layer that contains the sampled source.

Update Mode
  Specifies the type of update mode for the sampled source. Select Dynamic if the position of the sample line changes.

**Related procedures:**

■ Editing Sample Lines (page 1145)

# Section Style Dialog Box

Use this dialog box to define a section style, which controls the way a section is displayed in a drawing.

**See also:**

■ Creating and Editing Section Styles (page 1132)

# Information Tab (Section Style Dialog Box)

Use this tab to display administrative data and edit primary information about the section style, such as its name and description.

This tab also contains information about who created or modified the section style and the date of these events. The following fields can be modified:

Name
  Specifies the name of the selected section style.

Description
  Specifies the optional description of the selected section style.

# Display Tab (Section Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of section components, such as segments and points.

On the Display tab, you can use the ByLayer or ByBlock reference settings to set Color, Linetype, Lineweight, and Plot Style values. Reference settings are indirect and require some planning.

For more information, see Display Tab (Style Dialog Box) (page 1801).

# Summary Tab (Section Style Dialog Box)

Use this tab to edit the settings for the section style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Section View Bands Dialog Box

Use this dialog box to specify the source surfaces for data band annotation.

**List Of Bands**

Location

    Specifies the location in the section view of data band annotation.

Band Type

    Displays the type of data band specified in the Band Set properties.

Style

    Specifies the style for the band in the Band Set properties. Click  to open the Select Style dialog box (page 1805).

Surface 1

    Specifies a surface sampled by the sample line that supplies the data for the band, including any corridor surface.

Surface 2

    Specifies an additional surface sampled by the sample line that supplies the data for the band, including any corridor surface. Not used for Section Segment type data bands.

**Related procedures:**

- Creating/Editing Section Views (page 1153)
- Section View Bands (page 1140)

# Section View Band Set Dialog Box

Use this dialog box to create a set of data band styles for section views.

A band style set is useful when you want to apply the same set of data bands to a number of section views. After naming your band set on the Information tab, use the Bands tab to specify the bands included in the set and their location above or below the section view grid.

**See also:**

- Section View Bands (page 1140)

# Information Tab (Section View Band Set Dialog Box)

Use this tab to record basic information about the band set, such as its name and description.

This tab also contains information about who created or modified the section view style and the date of these events. The following fields can be modified:

Name

    Specifies the name of the band style set.

Description

    Specifies the optional description of the selected band style set.

# Bands Tab (Section View Band Set Dialog Box)

Use this tab to specify which data bands are included in the set, along with their styles and positions relative to the section view grid.

Band Type

Specifies the type of data band to add to the set.

Select Band Style

Specifies the style for the band.

To edit a style or create a new one, click ![icon] and select from the list of operations. For more information, see Select Style dialog box (page 1805) and Section Data Band Style dialog box (page 2267). To examine the details of an existing style, select the style name in the list. Click ![icon] . For more information, see the Style Detail dialog box (page 1804).

Add>>

Adds the specified data band type to the style set. Before clicking, ensure that the settings are correct for data band type, style, and location.

## List Of Bands

Location

Specifies either the top or bottom of the section view. For the current location, the table below this field shows the bands in the current band style set.

Band Type

Displays the data band type.

Style

Specifies the style for the band type.

Description

Specifies the optional description of the band style.

Gap

Specifies the distance in plotted units between the current data band and the previous data band (or the nearest section view axis).
For bands below the section view grid, the specified gap is measured from the top of the current band to the bottom of the previous band (or the section view bottom axis, if the current band is the first one). Similarly, for bands above the section view grid, the specified gap is measured from the bottom of the current band to the top of the previous band (or the section view top axis, if the current band is the first one).

Major Interval

Specifies the section view data band major interval distance.

Minor Interval

Specifies the section view data band minor interval distance.

Label End Offset

Specifies whether or not the end of the band is labeled.

Weeding

Used only for labels at vertical geometry points on Section Data bands. Vertical geometry points or grade breaks that are closer than the weeding factor are removed, making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance.

Stagger Labels

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left.

Stagger Line Height

Specifies the height of the label stagger line.

⇧

Moves the selected data band up in the list.

**NOTE** The order in this list is the order in which the bands appear below/above the section view.

⇩

Moves the selected data band down in the list.

✖

Deletes the selected data band from the list and removes it from the section view.

Match Major/Minor Increments to Vertical Grid Intervals

Specifies if the section data band major/minor intervals for section views matches the section view style's major/minor grid spacing interval distances. Select the checkbox to match major/minor increments to vertical grid intervals. If this checkbox is selected, the major/minor interval fields in the List of Bands table are unavailable.

# Section View Group Bands Dialog Box

Use this dialog box to edit properties that manage the last applied data bands for the section view group.

This dialog box is accessed from the Section View Group Properties dialog box and displays the last applied set of bands.

Band Type

Specifies which type of data band to add to the section view. Select a band type in the list: either Section Data or Section Segment.

Select Band Style

Specifies the style for the band. Select a band style in the list.

Creates, copies, or edits a style using the Section Data Band Style dialog box (page 2267). Click the down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.

Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Add>>

Adds the specified data band type to the set for this section view. Before clicking, ensure that the settings are correct for data band type, style, and location.

### List Of Bands

Location

Specifies the location where you want to draw the data band relative to the current section view.

- **Bottom Of Section View**: Draws the data band at the bottom (below) the current section view.

- **Top Of Section View**: Draws the data band at the top (above) the current section view.

Band Type

Displays the current data band type that was added using the Band Type list.

Style

Specifies the name of the band style that was added using the Select Band Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Description

Specifies the optional description of the band style.

Gap

Specifies the distance in plotted units between the current data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

- For bands below the section view grid, the specified gap is measured from the top of the current band to the bottom of the previous band (or the section view bottom axis, if the current band is the first one).

- For bands above the section view grid, the specified gap is measured from the bottom of the current band to the top of the previous band (or the section view top axis, if the current band is the first one).

Label Start Offset

Specifies whether or not the start end of the band is labeled.

Label End Offset

Specifies whether or not the end of the band is labeled.

Surface1/Surface2

Specifies two surfaces that are used by the band styles to annotate either elevations or the elevation differences between the two sections. Select the surface(s) by name in the list.

Weeding

Used only for labels at vertical geometry points on Section Data bands. Vertical geometry points or grade breaks that are closer than the weeding factor are removed, making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance.

Stagger Labels

Specifies the location of the data band label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left.

Stagger Line Height

Specifies the height of the data band label stagger line.

⬆

Moves the selected data band up in the list.

**NOTE** The order in this list is the order in which the bands appear below/above the section view.

Moves the selected data band down in the list.



Deletes the selected data band from the list and removes it from the section view.

Match Major/Minor Increments to Vertical Grid Intervals

Specifies if the section data band major/minor intervals for section views matches the section view style's major/minor grid spacing intervals. Select the checkbox to match major/minor increments to vertical grid intervals.

Import Band Set

Imports a band set. In the Band Set dialog box (page 1805), select an existing band set to be added to the current section view in the list.

Save As Band Set

Saves a new band set. Click to open the Section View Band Set dialog box (page 2275). Save the current list of bands as a band set for use with other section views.

**Related procedures:**

■ Adding Section View Bands (page 1141)

# Section View Group Properties Dialog Box

Use this dialog box to edit properties of the selected section view group, such as its name, the name of its parent alignment, properties of its constituent section views, and properties of the displayed sections.

In the Sections tab, you edit all options for the entire group.

In the Section Views tab, you can only edit section view names individually. You can edit section view styles individually or for the entire section view group. You can only set band set, volume tables and profile grade settings for the group.

You cannot edit station location, or the locations of Start and End stations.

**See also:**

■ Editing Section View Properties (page 1156)

# Section Views Tab (Section View Group Properties Dialog Box)

Use this tab to edit properties of the section views in the selected sample line group.

Sample Line Group Name

Specifies the name of the sample line group being resampled.

Alignment Name

Specifies the name of the parent alignment of the sample line group.

**Section View List**

Section View

Displays the names of section views arranged in an expandable tree or hierarchy.

**Group Plot Style**

Specifies the name of the plot style of the section view associated with the selected sample line group. Click in this column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

**Style**

Specifies the name of the style of the section view associated with the selected sample line group. Click in this column to open the Select Style dialog box. Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

**Change Band Set**

Displays the last applied set of bands for the current section view group. Click in this column to open the Section View Group Bands dialog box. Use this dialog box to edit properties that manage the data bands for the section view group.

**Section Display**

Specifies the section display options for the current section view. Click in this column to open the Section Display Controls dialog box (page 2269). Select section display options in the list.

**Profile Grade**

Specifies the grid lines displayed at offset alignments and other profiles within the range of section view extents. Click in this column to open the Profile Grade Points dialog box (page 2254).

**Station**

Displays the station value of the sample line along which this section view is created.

**Start Station**

Displays the beginning station in the range.

**End Station**

Displays the end station in the range.

## Sections Tab (Section View Group Properties Dialog Box)

Use this tab to edit section view display options.

### Edit Section Options Table

**Draw**

Specifies whether the section is drawn in the current section view. Select the check box to draw the section in the current section view.

**Clip Grid**

Specifies whether the grid lines in the current section view are clipped to the section (horizontal and/or vertical depending on the section view style definition). Select the radio button to clip the grid lines in the section view to the section.

**Change Labels**

Specifies the style set from which the section labels will be drawn. Click in this column to open the Select Style dialog box (page 1805).

**Style**

Specifies the section style. Click in this column to open the Select Section Style dialog box.

**Override Style**

Specifies the style for a section. Select the check box to open the Select Section Style dialog box.

# Section View Properties Dialog Box

Use this dialog box to edit properties of the selected section view, such as its name, grid dimensions, and properties of the displayed sections and data bands.

**See also:**

■ Editing Section View Properties (page 1156)

■ Creating and Editing Section Views (page 1152)

## Information Tab (Section View Properties Dialog Box)

Use this tab to edit primary information about the selected section view.

Name

Specifies the name of the section view. The default format is <start-station > (n).
<start-station> is the name of the beginning station in the section view.
n is an integer greater than or equal to one, which increments as section views are created in the drawing.

Description

Specifies the optional description of the section view.

Object Style

Specifies the name of the style used by the section view.

Creates, copies, or edits a style using the Section View Style dialog box (page 2288). Click the down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.

■ **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.

Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (not over toolbar icons).

## Offsets Tab (Section View Properties Dialog Box)

Use this tab to display the properties of the selected section view and the section view grid on which sections are displayed.

Sample Line

Displays the name of the parent sample line for the current section view.

Sample Line Group

Displays the name of the parent sample line group for the current section view.

Alignment

Displays the name of the parent horizontal alignment for the current section view.

**Offset Range**

Automatic

Specifies that the offset range is set automatically. The lengths of the left and right swath widths are displayed in the fields next to the selection.

User Specified

Specifies that the offset range is set by the user. Specify the lengths of the left and right swath widths in the fields next to the selection.

Left

Specifies the offset length of the left swath width.

Right

Specifies the offset length of the right swath width.

# Elevations Tab (Section View Properties Dialog Box)

Use this tab to view or specify the elevation properties of the selected section view and the section view grid on which sections are displayed.

**Elevation Range**

Automatic

Specifies that the elevation range is set automatically. The minimum and maximum heights are displayed in the fields next to the selection.

User Specified

Specifies that the elevation range is set by the user. Specify minimum and maximum heights in the fields next to the selection.

Minimum

Specifies the minimum height of the elevation.

Maximum

Specifies the maximum height of the elevation.

# Sections Tab (Section View Properties Dialog Box)

Use this tab to edit properties that determine which sampled sections are drawn in the section view.

Name

Displays the name of the current section.

Draw

Specifies whether the section is drawn in the current section view. Select the check box to draw the section in the current section view.

Clip Grid

Specifies whether the grid lines in the current section view are clipped to the section (horizontal and/or vertical depending on the section view style definition). Select the check box to clip the grid lines in the section view to the section.

Description

Specifies the optional description of the current section.

Type

Displays the type of data source from which the section data was extracted. Either ![icon], which indicates from an existing TIN surface; ![icon], which indicates from a corridor; ![icon], which indicates from a corridor surface; or ![icon], which indicates from an existing pipe network.

Data Source

Displays the name of the data source from which the current section is sampled.

Update Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface or corridor) geometry changes.

Layer

Specifies the drawing layer that contains the current section view associated with the selected sample line group. Click in this column to open the Layer Selection dialog box (page 1986). Select a layer in the Layers table.

Style

Specifies the name of the style of the section. Click in the Style column to open the Select Style dialog box (page 1805). Select a style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Override Style

Specifies whether the current style is overridden in the current section view. If the style is overridden, the cell is selected. The name of the override style is displayed.

Labels

Specifies the section label style. In the Labels column, click <Edit...> to open the Section Labels dialog box (page 2270). Select the label style.

Left Offset

Displays the left offset value of the current section.

Right Offset

Displays the right offset value of the current section.

Minimum Elevation

Displays the minimum elevation of the section associated with the selected sample line group.

Maximum Elevation

Displays the maximum elevation of the section associated with the selected sample line group.

Station

Displays the station value of the sample line along which the section is sampled.

## Bands Tab (Section View Properties Dialog Box)

Use this tab to edit properties that manage the data bands for the section view.

Band Type

Specifies which type of data band to add to the section view. Select a band type in the list: either Section Data or Section Segment.

Select Band Style

Specifies the style for the band. Select a band style in the list.

Creates, copies, or edits a style using the Section Data Band Style dialog box (page 2267). Click the down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.



Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

Add>>

Adds the specified data band type to the set for this section view. Before clicking, ensure that the settings are correct for data band type, style, and location.

**List Of Bands**

Location

Specifies the location where you want to draw the data band relative to the current section view.

■ **Bottom Of Section View**: Draws the data band at the bottom (below) the current section view.

■ **Top Of Section View**: Draws the data band at the top (above) the current section view.

Band Type

Displays the current data band type that was added using the Band Type list.

Style

Specifies the name of the band style that was added using the Select Band Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Description

Specifies the optional description of the band style.

Gap

Specifies the distance in plotted units between the current data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

■ For bands below the section view grid, the specified gap is measured from the top of the current band to the bottom of the previous band (or the section view bottom axis, if the current band is the first one).

■ For bands above the section view grid, the specified gap is measured from the bottom of the current band to the top of the previous band (or the section view top axis, if the current band is the first one).

Label Start Offset

Specifies whether or not the start end of the band is labeled.

Label End Offset

Specifies whether or not the end of the band is labeled.

Section1/Section2

Specifies two sections that are used by the band styles to annotate either elevations or the elevation differences between the two sections. Select the section(s) by name in the list.

Weeding

Used only for labels at vertical geometry points on Section Data bands. Vertical geometry points or grade breaks that are closer than the weeding factor are removed, making it easier to read the remaining labels. Enter a positive number to specify a label exclusion distance.

Stagger Labels

Specifies the location of the label staggering line: No staggering, Stagger both sides, Stagger to right, or Stagger to left.

Stagger Line Height

Specifies the height of the label stagger line.

⇧

Moves the selected data band up in the list.

NOTE The order in this list is the order in which the bands appear below/above the section view.

⇩

Moves the selected data band down in the list.

✖

Deletes the selected data band from the list and removes it from the section view.

Match Major/Minor Increments to Vertical Grid Intervals

Specifies if the section data band major/minor interval distances for section views matches the section view style's major/minor grid spacing intervals. Select the checkbox to match major/minor increments to vertical grid intervals. If this checkbox is selected, the major/minor interval fields are unavailable.

Import Band Set

Imports a band set. In the Band Set dialog box (page 1805), select an existing band set to be added to the current section view in the list.

Save As Band Set

Saves a new band set. Click to open the Section View Band Set dialog box (page 2275). Save the current list of bands as a band set for use with other section views.

**Related procedures:**

■ Adding Section View Bands (page 1141)

# Volume Tables Tab (Section View Properties Dialog Box)

Use this tab to set volume table properties for the section view.

Use the following controls to specify the volume table types to draw:

Type

Specifies the type of the volume table(s).

Select Table Style

Specifies the style of the volume table. Standard buttons are also provided to create or edit a label style.

Add >>

Adds a volume table to the list.

**List Of Volume Tables**

Table Type

Displays the current table type.

Style

Specifies the name of the table style that was added using the Select Table Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to create a new style, copy or edit the current style selection, or pick a style from the drawing.

Material List

Specifies the material list for the volume table.

Materials

Click ⋯ to specify the materials for the volume table.

Layer

Specifies the drawing layer that contains the sampled source.

Split

Specifies whether the layer is split.

Gap

Specifies the distance in plotted units between the current data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

Reactivity Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically if the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

**Position Of Table(s) Relative To Section View**

Section View Anchor

Specifies the section view anchor (Top/Middle/Bottom, Left/Center/Right).

Table Anchor

Specifies the location of the table anchor (Top/Middle/Bottom, Left/Center/Right).

Table Layout

Specifies whether the layout of the tables will be horizontal or vertical, in relation to the section view.

X Offset

Specifies the distance in plotted units to move the title away from the position set by the table anchor. Enter a positive number to move the title right, or a negative number to move it left.

Y Offset

Specifies the distance in plotted units to move the title away from the position set by the table anchor. Enter a positive number to move the title up, or a negative number to move it down.

**Related procedures:**

■ Creating/Editing Section Views (page 1153)

## Profile Grade Lines Tab (Section View Properties Dialog Box)

Use this tab to select a profile and display its elevation by marking it at the appropriate offset and elevation in the section view.

Alignment

Specifies the parent alignment whose profile grade point is to be displayed in this section view. Select an alignment from the drop-down list or click  to pick an alignment from the drawing.

Add>>

Adds the profile grade point for the currently selected alignment to the list.



Deletes the highlighted profile grade point from the list. The profile grade point will no longer be displayed in the section view.

### List Of Profile Grade Points

Alignment

Displays the name of the parent alignment whose profile grade point is marked in this section view.

Show

Specifies whether the marker is visible in the section view (that is, without having to edit the marker style or layer).

Profile

Displays the name of the profile whose location, with respect to the current section view, is marked.

Marker Style

Displays the marker style used for the profile grade point.

## Projections Tab (Section View Properties Dialog Box)

Use this tab to show or hide projected objects in a section view.

Name

Specifies the names and categories of objects that are available for display in the current section view. Click to add or remove objects in the section view.

Style

Specifies the display style of each projected object. Click a single entry to change it, or click <set all> for a category to select a style for all objects of that category.

Elevation Options

Specifies whether the elevation of the projected object is determined by its elevation in plan view or manually controlled in the section view. Click a single entry to change it, or click <set all> for a category to select an option for all objects of that category.

Elevation Value

Specifies the actual elevation of the projected object. Click <set all> for a category to select an elevation for all objects of that category.

Pick Objects

Click to select objects in the drawing and add them to the section view.

# Section View Style Dialog Box

Use this dialog box to define a section view style, which controls the way a section view is displayed in a drawing.

You can change the graph, grid, title, axes ticks, and text. Select which elements to display.

**See also:**

- Creating and Editing Section View Styles (page 1133)
- Creating and Editing Section Views (page 1152)

## Information Tab (Section View Style Dialog Box)

Use this tab to display administrative data and edit primary information about the section view style, such as its name and description.

This tab also contains information about who created or modified the section view style and the date of these events. The following fields can be modified:

Name

Specifies the name of the selected section view style.

Description

Specifies the optional description of the selected section view style.

## Graph Tab (Section View Style Dialog Box)

Use this tab to edit settings for vertical scale.

### Vertical Scale

Use the controls in this area to specify the elevation values, in order to improve visibility in the section view. Specify the section view vertical scale in either of the following ways:

- By Vertical Exaggeration — Enter a value in the Vertical Exaggeration field. The Vertical Scale value is calculated automatically.
  The Current Horizontal Scale (HS) factor of the drawing is divided by the Vertical Scale value (VS) to obtain the profile view Vertical Exaggeration value (VX).

$$\frac{HS}{VS} = VX$$

  For example, if the Current Horizontal Scale is 50, and you want the profile view's Vertical Scale to be 1"=10', then set the Vertical Exaggeration to 5.

$$\frac{50}{10} = 5$$

- By Vertical Scale — Select a scale from the Vertical Scale list. Alternatively, enter a custom value in the Custom field. The Vertical Exaggeration value will be calculated automatically.

Vertical Scale

The section view vertical scale. Select a vertical scale from the list, or enter a custom scale in the Custom Scale field.

Vertical Scale (VS) = Current Horizontal Scale (HS) divided by Vertical Exaggeration (VX)

$$VS = \frac{HS}{VX}$$

Custom Scale

Specifies the current Vertical Scale factor. Enter a value in this field to specify a scale factor that is not provided in the Vertical Scale list.

Current Horizontal Scale

The overall drawing Scale specified in the Drawing Settings dialog box (page 1851). This field is not editable. The horizontal scale must be changed in the Drawing Settings dialog box.

Vertical Exaggeration

Specifies by how much the elevation values increase in the section view for greater visibility. Enter either 1 for no increase in scale or a larger number to increase the scale. The elevation values are multiplied by this value, so a larger number increases the amount of exaggeration in the section view.

# Grid Tab (Section View Style Dialog Box)

Use this tab to specify clipping, padding, and axis offset options on the section view grid.

## Grid Options

 Clip Vertical Grid

Specifies that vertical grid lines are only drawn below the section line. If Clip To Highest Section(s) is not selected, the vertical lines will be drawn only below the section view line that has Clip Grid selected on the Sections tab (page 2282) in the Section View Properties dialog box.

**Clip To Highest Section(s)**: Specifies that vertical section view grid lines will be drawn to the highest of all displayed profiles.

**Omit Grid In Padding Areas**: Specifies that vertical profile view grid lines will not be drawn in the padding areas.

 Clip Horizontal Grid

Specifies that horizontal grid lines are only drawn below the section line. If Clip To Highest Section(s) is not selected, the horizontal lines will be drawn only below the section line that has Clip Grid selected on the Sections tab in the Section View Properties dialog box.

**Clip To Highest Section(s)**: Specifies that vertical section view grid lines will be drawn to the highest of all displayed profiles.

**Omit Grid In Padding Areas**: Specifies that horizontal section view grid lines will not be drawn in the padding areas.

## Grid Padding

 Grid Padding (Major Grids)

Specifies the distance (in major grid blocks) to add to the extents of the section view in relation to the profiles it contains.

## Above Maximum Elevation

Specifies the number of grid blocks that the section view extends above the maximum section elevation. This enables the graph to be taller than the extents of the section drawn in the profile view.

## Below Datum

Specifies the number of grid blocks that the section view extends below the datum or minimum section elevation. This enables the graph to extend below the extents of the section drawn in section view.

## To Left

Specifies the number of major grids to place before the left-most station value of the alignment. This enables the graph to be wider than the extents of the section drawn in the section view.

## To Right

Specifies the number of major grids to place after the right-most station value of the alignment. This enables the graph to be wider than the extents of the section drawn in the section view.

### Axis Offset

**NOTE** The axis offset distance is specified in plotted units, based on the Drawing Units and Scale settings specified on the Units and Zone tab (page 1851) of the Drawing Settings dialog box.

 Axis Offset (Plotted Units)

Specifies the distance (in plotted units) to offset the section view's horizontal and vertical axes from the grid extents (including padding). This creates a blank area between the axis and the grid.

## Above Maximum Elevation

Specifies the distance to offset the horizontal axis above the vertical extent.

## Below Datum

Specifies the distance to offset the horizontal axis below the vertical extent.

## To Left

Specifies the distance to offset the horizontal axis to the left of the horizontal extent.

## To Right

Specifies the distance to offset the horizontal axis to the right of the horizontal extent.

# Title Annotation Tab (Section View Style Dialog Box)

Use this tab to specify settings for the graph view and axis titles.

### Graph View Title

Text Style

Specifies the text style for the section view title.

Text Height

Specifies the current text height in drawing units.

Title Content

Specifies the text elements used to compose the title, such as horizontal alignment (HA) name and station

range. To change title format or content, click  to open the Text Component Editor dialog box (page 1961).

## Title Position

These controls specify the position of the profile view title. Use Location and Justification settings for basic positioning, then X and Y offsets for more precise placement, if necessary.

Location

Specifies the location of the title in relation to the section view grid: either top, bottom, left, or right.

Justification

Specifies the justification (alignment) of the title. If Location is top or bottom, justification values are left, right, or center. If Location is right or left, justification values are top, middle, or bottom.

X Offset

Specifies a horizontal offset from the position set by the Location and Justification values. Enter a positive number to move the title right or a negative number to move it left.

Y Offset

Specifies a vertical offset from the position set by Location and Justification values. Enter a positive number to move the title up or a negative number to move it down.

Border Around The Title

Specifies whether a border line is drawn around the title block.

Gap

Specifies the distance between the title and its border.

## Axis Title Text

Separate titles may be displayed at each of the axes. Use the radio buttons around the graphic to select the axis title to edit. Configure title settings and click Apply. Repeat that process with each axis title.

NOTE The Axis Title selections do not control axis title visibility. Visibility controls for all section components are located on the Display tab (page 2295).

 Axis Title Text

Specifies which axis title is being controlled by the text editing tools.

Title Text

Specifies the title text for the horizontal axis. To change the title content, click  to open the Text Component Editor dialog box (page 1961).

Location

Specifies the location of the title on the axis: either Left, Center, or Right. For more precise control, use the X and Y Offsets.

Rotation

Specifies the angle for the title. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the title is not rotated.

Text Style

Specifies a text style for use in the horizontal axis title.

Text Height

Specifies the title text height (in plotted units).

X Offset

Specifies a horizontal offset from the Location setting. Enter a positive number to move the title right or a negative number to move it left.

Y Offset

Specifies a vertical offset from the Location setting. Enter a positive number to move the title up or a negative number to move it down.

## Horizontal Axes Tab (Section View Style Dialog Box)

Use this tab to edit settings for the section view style, such as the title format, use of tick marks, and annotations on the horizontal axes of the section view.

After you configure horizontal axis title and tick details, click Apply. Later, you can change the Select Axis setting and configure another axis on the opposite side of the grid.

Select Axis To Control

Click either the Top or Bottom radio button to select that axis.

### Major Tick Details

Interval

Specifies the spacing of major ticks on the horizontal axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified Top, Bottom or Center.

Text Height

Specifies the height of text used to label major ticks on the horizontal axis, using in plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box (page 1961).

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

### Minor Tick Details

Interval

Specifies the spacing of minor ticks on the horizontal axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified Top, Bottom or Center.

Text Height

Specifies the height of text used to label minor ticks on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

## Horizontal Geometry Tick Details

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified Top, Bottom or Center.

Text Height

Specifies the height of text used to label minor ticks on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

# Vertical Axes Tab (Section View Style Dialog Box)

Use this tab to edit settings for the section view style, such as the title location and format, use of tick marks, and annotations, on the vertical axis of the section view.

After you configure vertical axis title and tick details, click Apply. Later, you can change the Select Axis setting and configure another axis on the opposite side of the grid.

Select Axis To Control

Click either the Left, Center or Right radio button to select that axis.

## Major Tick Details

Interval

Specifies the spacing of major ticks on the vertical axis. Enter a positive value in ground units.

Tick Size

Specifies the length of the actual tick on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified Left, Right or Center.

Text Height

Specifies the height of text used to label major ticks on the vertical axis in plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box (page 1961).

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the end of the tick.

Y Offset

Specifies a vertical offset for the tick label from the end of the tick.

## Minor Tick Details

Interval

Specifies the spacing of minor ticks on the vertical axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the length of the actual tick on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified Left, Right or Center.

Text Height

Specifies the height of text used to label minor ticks on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. To change the text format or content, click ![icon] to display the Text Component Editor dialog box.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the end of the tick.

Y Offset

Specifies a vertical offset for the tick label from the end of the tick.

# Display Tab (Section View Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of section view components, such as titles, axes annotations, tick marks, and grid options.

For more information, see Display Tab (Style Dialog Box) (page 1801).

# Summary Tab (Section View Style Dialog Box)

Use the Summary tab to edit settings for the section style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Select Page Layout Dialog Box

Use this dialog box to select a page layout for the sheet style.

Specifies the default page layout format. Select a page layout format from the list. Click OK.

## Page Setups

Default (Layout)

Plots section views on a sheet using the Default (Layout) format.

Default (Model)

Plot section views on a sheet using the Default (Model) format.

For more information about the sheet style, see Sheet Style Dialog Box (page 2296).

# Sheet Properties Dialog Box

Use this dialog box to edit properties of a sheet used for multiple section views.

Select an object style; and create, copy, edit or pick a sheet style for your section view sheets.

Name

Specifies the name of the sheet.

Description

Specifies the optional description of the sample line.

Object Style

Specifies the name of the style used by the sheet. Select an object style in the list.



Creates, copies, or edits a style using the Sheet Style dialog box (page 2296). Click the down arrow to display the style selection menu:

- **Create New**: Creates a new style.

- **Copy Current Selection**: Copies the current style.

- **Edit Current Selection**: Edits the current style.

- **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



Displays details about an existing style. Select the style by name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

**Related procedures:**

- Creating and Editing Section Views (page 1152)

# Sheet Style Dialog Box

Use this dialog box to edit settings that define a sheet style, which controls the way a section sheet is displayed in a drawing.

**See also:**

- Creating and Editing Section View Styles (page 1133)

## Information Tab (Sheet Style Dialog Box)

Use this tab to display administrative data and edit primary information about the sheet style, such as its name and description.

Name

Specifies the name of the selected sheet style.

Description

Specifies the optional description of the selected sheet style.

## Sheet Tab (Sheet Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of sheet components, such as page layout, margins, and plot area grid details.

Page Layout

Specifies the default page layout format. Select a page layout format from the list.

**Page Margins**

Top

Specifies the top page margin in plotted units. Enter a positive value in plotted units.

Bottom

Specifies the bottom page margin in plotted units. Enter a positive value in plotted units.

Left

Specifies the left page margin in plotted units. Enter a positive value in plotted units.

Right

Specifies the right page margin in plotted units. Enter a positive value in plotted units.

**Plot Area Grid Details**

Horizontal Major

Specifies the horizontal major grid line interval in plotted units. Enter a positive value in plotted units.

Horizontal Minor

Specifies the horizontal minor grid line interval in plotted units. Enter a positive value in plotted units.

Vertical Major

Specifies the vertical major grid line interval in plotted units. Enter a positive value in plotted units.

Vertical Minor

Specifies the vertical minor grid line interval in plotted units. Enter a positive value in plotted units.

## Display Tab (Sheet Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of sheet components, such as grids, print area, and sheet border.

For more information, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Sheet Style Dialog Box)

Use this tab to edit settings for the sheet style.

To edit settings, click the Page Layout cell. Click  to open the Select Page Layout dialog box (page 2295).

For more information, see Summary Tab (Style Dialog Box) (page 1803).

# Material Analysis Dialog Boxes

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The following topics provide information about the Quantity Takeoff and Mass Haul dialog boxes.

## Change Volume Table Dialog Box

Use this dialog box to change volume table(s) in the drawing.

Use the following controls to specify the volume table types to draw:

Type
  Specifies the type of the volume table(s).

Select Table Style
  Specifies the style of the volume table. Standard buttons are also provided to create or edit a label style.

Add >>
  Adds a volume table to the list.

### List Of Volume Tables

Table Type
  Displays the current table type.

Style
  Specifies the name of the table style that was added using the Select Table Style list. To change the style, click in this column to open the Select Style dialog box (page 1805). Select a band style in the list or use the standard controls to either create a new style, copy or edit the current style selection, or pick a style from the drawing.

Material List
  Specifies the material list for the volume table.

Materials
  Click  to specify the materials for the volume table.

Layer
  Specifies the drawing layer that contains the sampled source.

Split

Specifies whether the layer is split.

Gap

Specifies the distance in plotted units between the current data band and the previous data band (or the nearest section view axis). Enter a positive gap value in plotted units.

Reactivity Mode

Specifies the type of update mode for the section. Select Dynamic to specify that the section data updates dynamically when either the position of the sample line associated with the section changes or the data source (surface, corridor, or pipe) geometry changes.

**Position Of Table(s) Relative To Section View**

Section View Anchor

Specifies the section view anchor (Top/Middle/Bottom, Left/Center/Right).

Table Anchor

Specifies the location of the table anchor (Top/Middle/Bottom, Left/Center/Right).

Table Layout

Specifies whether the layout of the tables is horizontal or vertical, in relation to the section view.

X Offset

Specifies the distance, in plotted units, to move the title away from the position set by the table anchor. Enter a positive number to move the title right, or a negative number to move it left.

Y Offset

Specifies the distance, in plotted units, to move the title away from the position set by the table anchor. Enter a positive number to move the title up, or a negative number to move it down.

**Related procedures:**

■  Creating/Editing Section Views (page 1153)

# Compute Materials Dialog Box

Use this dialog box to create material lists for a sample line group.

Quantity Takeoff Criteria

Specifies the criteria to use. Select from the list or click  to edit or create a criteria.

Curve Correction Tolerance

Specifies whether curve tolerance is used. If selected, enter a value for the curve tolerance in the adjacent field.

Map Objects With Same Name

Maps criteria surface or corridor shape names with surface or corridor shape names in the drawing that are the same.

The Properties table contains the following columns:

Name In Criteria

Displays the surface and corridor shape names that are used in the criteria.

Object Name

Specifies the actual object name to map to the surface or corridor shape name in the criteria.

Material Name

Displays the material name that is referenced by the corresponding surface or corridor shape in the list.

**Related procedures:**

■ Analyzing Sectional Volumes (page 1168)

# Create Mass Haul Diagram Wizard

Use this wizard to create mass haul diagrams.

---

**NOTE** A material list must be created for the drawing before a mass haul diagram can be created. For information about how to create a material list, see Generating Material Lists (page 1163).

---

## General Page (Create Mass Haul Diagram Wizard)

Use this page to specify basic information about the mass haul diagram, including the parent alignment and sample line group, mass haul view name, description, style, and layer.

Select Alignment

Specifies the name of the parent horizontal alignment. Select a name from the list or click ![icon] to pick an alignment from the drawing.

Sample Line Group

Specifies the parent sample line group. Select a name from the list or click ![icon] to pick an alignment from the drawing.

Mass Haul View Name

Specifies a system-generated name that you can edit. The name must remain unique within the drawing.

Click ![icon] to open the Name Template dialog box (page 1806), where you can modify the material naming template.

Description

Specifies an optional description of the mass haul diagram.

Mass Haul View Style

Specifies the style for the mass haul view. Use the standard controls to review or change the style.

Mass Haul View Layer

Specifies the drawing layer for the mass haul view. To review or change layer data, click ![icon] .

**Related procedures:**

■ Creating Mass Haul Diagrams (page 1174)

## Mass Haul Display Options Page (Create Mass Haul Diagram Wizard)

Use this page to specify mass haul diagram display options.

### Material

Material List

Specifies the material list for the mass haul diagram.

Choose A Material To Display As Mass Haul Line

Specifies the material to display as the mass haul line. Select from Total Volume, Total Cut Volume, Total Fill Volume, Total Unusable Volume, and materials specific to the material list you have selected.

**Mass Haul Line**

Mass Haul Line Name

Specifies a system-generated name that you can edit. The name must remain unique within the drawing.

Description

Specifies an optional description of the mass haul diagram.

Mass Haul Line Style

Specifies the style for the mass haul line. Use the standard controls to review or change the style.

Mass Haul Line Layer

Specifies the drawing layer for the mass haul line. To review or change layer data, click  .

**Related procedures:**

■ Creating Mass Haul Diagrams (page 1174)

# Balancing Options Page (Create Mass Haul Diagram Wizard)

Use this page to set free haul distance, and borrow pit/dump site properties for the mass haul diagram.

Use the following controls to specify the free haul distance and type, location and capacity of borrow pits/dump sites:

**Free Haul Options**

Free Haul Distance

Select this option to set a free haul distance value. If the option is cleared, no free haul distance is assigned to the mass haul diagram. If you select the option, you can either accept the default or set your own free haul distance in the field below.

**Add/Remove Borrow Pits And Dump Sites**

Add Borrow Pit

Click to add a borrow pit to the list.

Add Dump Site

Click to add a dump site to the list.



Deletes any selected borrow pit or dump site from the list.

**List**

Type

Specifies whether the balancing feature is a borrow pit or a dump site. Click in this field to change the type of balancing feature.

Station

Specifies the station at which the borrow pit or dump site will be located. The station can be specified either by entering a station value or selecting the station in the viewport. When you select the station in the viewport, you must select it from a point along the alignment in the plan view, and not along the X axis of the mass haul view.

Capacity
>Specifies the volume to be borrowed or dumped at the borrow pit or dump site.

**Related procedures:**

■ Creating Mass Haul Diagrams (page 1174)

# Create Material Volume Table Dialog Box

Use this dialog box to create a table in the drawing that contains volume data for a specific material in a material list.

Most of the controls in this dialog box are generic table creation controls. For more information, see Table Creation Dialog Box (page 2441).

Use the following controls to specify the materials information to be included in the material volume table:

Select Alignment

>Specifies the alignment. Select an alignment from the list or click  to select an alignment in the drawing. The alignments in the list are associated with at least one sample line group.

Select Sample Line Group
>Specifies the sample line group to use. Select a group from the list.

Select Material List
>Specifies which material list defined in the sample line group properties is used.

Select A Material
>Specifies which material from the material list is used to generate the table.

**Related procedures:**

■ Analyzing Sectional Volumes (page 1168)

# Create Total Volume Table Dialog Box

Use this dialog box to create a table in the drawing using volume data from material list for a sample line group.

Most of the controls in this dialog box are generic table creation controls. For a description of these controls, see Table Creation Dialog Box (page 2441).

Use the following controls to specify the materials information to be included in the total volume table:

Select Alignment

>Specifies the alignment. Select an alignment from the list or click  to select an alignment in the drawing. The alignments in the list are associated with at least one sample line group.

Select Sample Line Group
>Specifies the sample line group to use. Select a group from the list.

Select Material List
>Specifies which material list defined in the sample line group properties is used.

**Related procedures:**

■ Analyzing Sectional Volumes (page 1168)

# Define Material Criteria Dialog Box

Use this dialog box to define quantity takeoff criteria from a sample line group.

Select Alignment

Specifies the alignment. Click  to select an alignment in the drawing.

Select Sample Line Group

Specifies the sample line group to use. Select a group from the list.

Select Surfaces

Displays the available surfaces from which sections are to be cut. Select the check box next to surface to use it to define a criteria material.

**NOTE** At least two surfaces must be selected. Surfaces selected are compared at each station and the enclosed areas at each cross section are added as a material type to the criteria.

**Related procedures:**

■ Creating Quantity Takeoff Criteria (page 1165)

# Edit Feature Settings - Mass Haul Line Dialog Box

Use this dialog box to view and change standard drawing ambient settings for mass haul line.

The standard ambient settings are preceded with  . For feature-specific Mass Haul Line Style and Mass Haul Line Name Template information, see Edit Feature Settings - Mass Haul View Dialog Box (page 2304).

For more information about this dialog box, see Working with the Standard Settings Dialog Box Controls (page 62).

**Related procedures:**

■ Editing Mass Haul Line Settings (page 1178)

# Edit Feature Settings - Mass Haul View Dialog Box

Use this dialog box to view and change mass haul view settings.

This topic documents settings in all mass haul-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the  drawing icon.

■ Mass haul feature settings are listed near the top of this dialog box, after the General property group, and are identified by the  mass haul view icon.

■ Mass haul command settings are identified by the  command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### ⌁ Default Styles

Use these settings to specify the default styles for mass haul components.

Mass Haul Line Style

Specifies the default style used for mass haul lines. Click in the Value column, and click ⊡ to select a style in the Mass Haul Line Style dialog box.

Mass Haul View Style

Specifies the default style used for mass haul views. Click in the Value column, and click ⊡ to select a style in the Mass Haul View Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

### ⌁ Default Name Format

Use these settings to specify the default name format for new mass haul lines and new mass haul views.

Click in the Value column, and click ⊡ to make changes in the Name Template dialog box (page 1806).

### ⌁ Mass Haul Creation

Use these standard table settings, which are available when accessing the settings from the CreateMassHaul command, to establish the default settings when you create a mass haul diagram.

Analyze Free Haul
Specifies whether the free haul distance is analyzed.

Free Haul Distance
Specifies the free haul distance.

Borrow Pit Capacity
Specifies the capacity of a borrow pit.

Dump Site Capacity
Specifies the capacity of a dump site.

**Related procedures:**

■ Editing Mass Haul View Settings (page 1179)

## Edit Feature Settings - Quantity Takeoff Dialog Box

Use this dialog box to view and change quantity takeoff-related settings.

This topic documents settings in all quantity takeoff-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

■ Drawing-level ambient settings are identified by the ⬚ drawing icon.

■ Quantity takeoff feature settings are listed near the top of this dialog box, after the General property group, and are identified by the ⊞ quantity takeoff icon.

■ Quantity takeoff command settings are identified by the ⊞ command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

## Default Styles

Use this setting to establish the default quantity takeoff criteria and styles assigned to quantity takeoff components.

Quantity Takeoff Criteria
Specifies the default quantity takeoff criteria when creating a quantity takeoff report. Click in the Value column, and click ⋯ to select criteria in the Quantity Takeoff Criteria dialog box.

Material Shape Style
Specifies the default style used for a material section when it is displayed in a section view. Click in the Value column, and click ⋯ to select a style in the Material Shape Style dialog box.

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use these settings to specify the default name format for new quantity takeoff material lists and materials. Click in the Value column, and click ⋯ to make changes in the Name Template dialog box (page 1806).

## Table Creation

**NOTE** This property group is displayed when you access the settings from the AddMaterialVolumeTable and AddTotalVolumeTable commands.

Use these settings to establish the defaults when you add a volume table.

Table Style

Specifies the style for a table. Click in the Value column, and click ⋯ to select a style in the Table Style dialog box.

Split Table
Specifies whether a table is split into two or more sections after a specified maximum number of rows has been met.

Maximum Number of Rows
Specifies the maximum number of rows to include per section. If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack
Specifies the maximum number of sections to include in each stack.

Table Spacing
Specifies the spacing between tables.

Tile Direction
Specifies the direction in which the table tiles (across or down).

## Define Materials Options

**NOTE** This property group is displayed when you access the settings from the ComputeMaterials and GenerateQuantitesReport commands.

Use these settings to establish the default settings when you create a new quantity takeoff report.

Apply Curve Correction

Specifies whether curve correction is used when the Curve Correction Tolerance value is exceeded. Curve correction may be required to correct volume calculations within a curvilinear section of an alignment (or on two sides of a horizontal intersection point).

Curve Correction Tolerance

Specifies the curve tolerance value. If the sample lines are within the tolerance or if the angle of deflection between the successive stations is small, then curve correction is not used. Enter an angle in degrees in the Value column or click  and select an angle in the drawing area.

 **Quantity Report Options**

**NOTE** This property group is displayed when you access the settings from the GenerateQuantitesReport command.

Use these settings to establish the default settings when you create a new quantity takeoff report.

Display XML Report

Specifies whether XML reports are displayed by default when you generate a quantity takeoff report.

# Edit Material List Dialog Box

Use this dialog box to review the material lists for a sample line group and add or delete materials from a list.

Add New Material

Adds a new empty material to a list. After you add the material, you can edit its type and settings and use the Define Material fields to populate it with data.



Opens the Name Template dialog box (page 1806), where you can modify the material naming template.

**Define Material**

Data Type

Specifies the type of data that is compared and processed when defining the material. Either Surface or Corridor Shape.

Select Surface/Corridor Shape

Lists the available surfaces or corridor shapes in the drawing. Select a surface or corridor shape or enter a surface or corridor shape name.

**NOTE** Corridor shapes can be added only to a material with a Structure quantity type.



Adds the data specified in the Define Material fields to the selected material name.

**NOTE** If a material is not selected, the data is not added.



Deletes the material or data component that is currently selected in the Material Name column.

**Material List**

The properties table contains the following columns:

**Material Name**

Specifies the material lists and names. Each instance of applying criteria to materials in a sample line group is added as a numbered list. List names can be edited.

Click ⊞ next to a list name to display its components (materials).Click ⊞ next to a material name to display its components (surfaces or structures).

**Condition**

Specifies the condition on which to base the calculation:

- **Above**. Specifies that an area above this surface is included in the material definition. Used with Below to define two or more surfaces for cut, fill, and structures material types.

- **Below**. Specifies that an area below this surface is included in the material definition. Used with Above to define two or more surfaces for cut, fill, and structures material types.

- **Base**. Specifies that this surface is the surface to compare against the compare surface. Used with Compare to define two or more surfaces for earthworks or cut and fill material types.

- **Compare**. Specifies that this surface is the surface to compare against the base surface. Used with Base to define two or more surfaces for earthworks or cut and fill material types.

- **Include**. Specifies a corridor shape that is included in the structure type definition.

**Quantity Type**

Specifies the quantity type:

- **Cut**. Calculates the material to remove.

- **Fill**. Calculates the material to add.

- **Cut And Refill**. Defines an area in the section where a material is removed and refilled with fill material.

- **Earthworks**. Compares two surfaces to calculate both cut and fill areas and displays them separately.

- **Structures**. Calculates the volume of one or more corridor shapes (as defined by the shape codes that are used to define the corridor). For information about shape codes, see Understanding Point, Link, and Shape Codes (page 1406).

**Cut Factor**

Specifies the expansion or swell of the cut material.

**Fill Factor**

Specifies the contraction or shrinkage of the fill material.

**Refill Factor**

Specifies usability factor used to calculate how much cut material can be reused as fill.

**Shape Style**

Specifies the default style used to display the material in a section view.

**Curve Tolerance**

Specifies the curve correction tolerance, if any, that was specified when the criteria was applied. To edit the value, select the check box, and then edit the value. Used only for List entries.

**Import Another Criteria**

Prompts you to define a new materials list for the sample line group. Opens the Compute Materials dialog box after you select a criteria.

**Related procedures:**

- Generating Material Lists (page 1163)

# Mass Haul Line Properties Dialog Box

Use this dialog box to edit mass haul line properties.

Mass haul line information can be viewed, but not edited, from the Mass Haul Line Data Tab (Mass Haul Line Properties Dialog Box).

## Information Tab (Mass Haul Line Properties Dialog Box)

Use this tab to edit the name, description, or object style of a mass haul diagram.

Name
  Specifies the name of the mass haul line.

Description
  Specifies the optional description of the mass haul line.

Object Style
  Specifies the name of the style used by the mass haul line.



  Creates, copies, or edits a style using the Mass Haul Line Style dialog box. Click the Down arrow to display the style selection menu:

  ■ **Create New**: Creates a new style.

  ■ **Copy Current Selection**: Copies the current style.

  ■ **Edit Current Selection**: Edits the current style.

  ■ **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



  Displays details about an existing style. Select the style name in the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

**Related procedures:**

■ Editing Mass Haul Line Properties (page 1175)

## Mass Haul Line Data Tab (Mass Haul Line Properties Dialog Box)

Use this tab to specify mass haul line data.

Material List Name
  Displays the name of the material list for the mass haul line.

Material Name
  Displays the material name that is referenced by the mass haul line.

Alignment Name
  Displays the name of the parent alignment of the mass haul line.

Sample Line Group

Displays the name of the parent sample line group.

**Related procedures:**

■ Editing Mass Haul Line Properties (page 1175)

## Balancing Options Tab (Mass Haul Line Properties Dialog Box)

Use this page to set free haul distance, and borrow pit/dump site properties for the mass haul diagram.

Use the following controls to specify the free haul distance and type, location, and capacity of borrow pits/dump sites:

### Free Haul Options

Free Haul Distance

Select this option to enter a free haul distance value in the field below. If the option is cleared, the free haul distance is set to its default.

### Add/Remove Borrow Pits and Dump Sites

Add Borrow Pit

Click to add a borrow pit to the list.

Add Dump Site

Click to add a dump site to the list.

✖

Deletes any selected borrow pit or dump site from the list.

### List

Type

Specifies whether the balancing feature is a borrow pit or a dump site. Click in this field to change the type of balancing feature.

Station

Specifies the station at which the borrow pit or dump site will be located.

Capacity

Specifies the volume either to be borrowed or dumped at the borrow pit or dump site.

**Related procedures:**

■ Editing Mass Haul Line Properties (page 1175)

## Mass Haul Line Style Dialog Box

Use this dialog box to define a mass haul line style, which controls the way a mass haul line is displayed in a drawing.

On the Display tab, you can use the ByLayer or ByBlock reference settings to set Color, Linetype, Lineweight, and Plot Style values. Reference settings are indirect and require some planning.

# Information Tab (Mass Haul Line Style Dialog Box)

Use this tab to display administrative data and edit primary information about the mass haul line style, such as its name and description.

This tab also contains information about who created or modified the mass haul line style and the date the style was created or modified. The following fields can be modified:

Name

Specifies the name of the selected mass haul line style.

Description

Specifies an optional description of the selected mass haul line style.

**Related procedures:**

■ Editing Mass Haul Line Styles (page 1177)

# Free Haul Tab (Mass Haul Line Style Dialog Box)

Use this tab to specify how you want to show free haul in graph.

Use the following controls to specify how free haul is graphed:

**Free Haul Options**



Measure From Grade Point

Select this option to show the free haul distance as measured from grade point in the mass haul diagram.



Measure From Balance Point

Select this option to show the free haul distance as measured from balance point in the mass haul diagram.

**Related procedures:**

■ Editing Mass Haul Line Styles (page 1177)

# Display Tab (Mass Haul Line Style Dialog Box)

Use this tab to edit settings for the visibility and format styles of mass haul line components, such as mass haul and free haul lines, and mass haul and free haul area hatch.

For more information, see Display Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■ Editing Mass Haul Line Styles (page 1177)

## Summary Tab (Mass Haul Line Style Dialog Box)

Use this tab to edit settings for the mass haul line style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

**Related procedures:**

■ Editing Mass Haul Line Styles (page 1177)

# Mass Haul View Properties Dialog Box

Use this dialog box to edit mass haul view properties.

Mass haul line information can be viewed, but not edited, from the Mass Haul Line Data Tab (Mass Haul View Properties Dialog Box).

# Information Tab (Mass Haul View Properties Dialog Box)

Use this tab to edit the name, description, or object style of a mass haul view.

Name
   Specifies the name of the mass haul view.

Description
   Specifies the optional description of the mass haul view.

Object Style
   Specifies the name of the style used by the mass haul view.



   Creates, copies, or edits a style using the Mass Haul View Style dialog box. Click the Down arrow to display the style selection menu:

   ■ **Create New**: Creates a new style.

   ■ **Copy Current Selection**: Copies the current style.

   ■ **Edit Current Selection**: Edits the current style.

   ■ **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



   Displays details about an existing style. Select the style name from the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

**Related procedures:**

■ Editing Mass Haul View Properties (page 1176)

# Mass Haul Line Tab (Mass Haul View Properties Dialog Box)

Use this tab to specify mass haul line data.

Mass Haul Line Name
   Displays the name of the mass haul line.

Alignment Name

Displays the name of the parent horizontal alignment.

Sample Line Group

Displays the name of the parent sample line group.

Material List Name

Displays the name of the material list for the mass haul line.

Material Name

Displays the material name that is referenced by the mass haul line.

**Mass Haul Line Display Options**

Mass Haul Line Display Style In This View

Specifies the name of the style used by the mass haul line in this view.



Creates, copies, or edits a style using the Mass Haul Line Style dialog box. Click the Down arrow to display the style selection menu:

■ **Create New**: Creates a new style.

■ **Copy Current Selection**: Copies the current style.

■ **Edit Current Selection**: Edits the current style.

■ **Pick From Drawing**: Prompts you to select a style directly from the drawing. The selected style becomes the current style.



Displays details about an existing style. Select the style name from the list. Click to open the Style Detail dialog box (page 1804). Preview the style and creation information.

**Related procedures:**

■ Editing Mass Haul View Properties (page 1176)

# Mass Haul View Style Dialog Box

Use this dialog box to define a mass haul view style, which controls the way a mass haul view is displayed in a drawing.

You can change the mass haul view graph, grid, title, axes ticks, and text. Select which elements to display.

# Information Tab (Mass Haul View Style Dialog Box)

Use this tab to display administrative data and edit primary information about the mass haul view style, such as its name and description.

This tab also contains information about who created or modified the mass haul view style and the date of these events. The following fields can be modified:

Name

Specifies the name of the selected mass haul view style.

Description

Specifies the optional description of the selected mass haul view style.

**Related procedures:**

- Editing Mass Haul View Styles (page 1178)

# Graph Tab (Mass Haul View Style Dialog Box)

Use this tab to edit settings for vertical scale.

### Vertical Scale

Vertical Scale

Specifies the vertical scale ratio. Standard ratios run from 1″ = 1′ to 1″ = 100′. Specifying a number in the Custom Scale field sets the scale to Custom.

Custom Scale

Specifies a custom vertical scale ratio.

Current Horizontal Scale

Displays the current horizontal scale from drawing settings.

Vertical Exaggeration

Displays a value specifying the increase in the elevation values in the mass haul view. The vertical exaggeration is equal to the current horizontal scale divided by the vertical scale.

### Mass Haul View Direction

Specifies the direction of the mass haul line for the mass haul view grid:



**Left To Right**

The mass haul view is drawn with either 0 or the lowest station number on the left the highest station number on the right.



**Right To Left**

The mass haul view is drawn with the highest station number on the left and either 0 or the lowest station number on the right.

**Related procedures:**

- Editing Mass Haul View Styles (page 1178)

# Grid Tab (Mass Haul View Style Dialog Box)

Use this tab to specify clipping, padding, and axis offset options on the mass haul view grid.

## Grid Options



Clip Vertical Grid

Specifies that vertical grid lines are drawn only between the mass haul line and the balancing line. Select **Omit Grid In Padding Areas** to specify that vertical grid lines are not drawn in the padding areas.



Clip Horizontal Grid

Specifies that horizontal grid lines are drawn only between the mass haul line and the balancing line. Select **Omit Grid In Padding Areas** to specify that horizontal grid lines are not drawn in the padding areas.

## Grid Padding



Grid Padding (Major Grids)

Specifies a value (in major grid blocks) that adds to the extents of the mass haul view in relation to the mass haul line it contains. This creates a graph that is taller than the extents of the mass haul line drawn in the mass haul view.

Above Maximum Point On Graph

Specifies the number of grid blocks that extend the mass haul view above the maximum point on the graph. This creates a graph that is taller than the extents of the mass haul line.

Below Minimum Point On Graph

Specifies the number of grid blocks that extend the mass haul view below the minimum point on the graph. This creates a graph that extends lower than the extents of the mass haul line.

To Left

Specifies the number of major grids to place before the left-most station value of the alignment. This creates a graph that is wider than the extents of the mass haul line.

To Right

Specifies the number of major grids to place after the right-most station value of the alignment. This creates a graph that is wider than the extents of the mass haul line.

## Axis Offset

NOTE  The axis offset distance is specified in plotted units, based on the Drawing Units and Scale settings specified on the Units and Zone tab (page 1851) of the Drawing Settings dialog box.



Axis Offset (Plotted Units)

Specifies the distance (in plotted units) to offset the mass haul view's horizontal and vertical axes from the grid extents (including padding). This creates a blank area between the axis and the grid.

Above Maximum Point On Graph

Specifies the distance to offset the horizontal axis above the vertical extent.

**Below Minimum Point On Graph**

Specifies the distance to offset the horizontal axis below the vertical extent.

**To Left**

Specifies the distance to offset the horizontal axis to the left of the horizontal extent.

**To Right**

Specifies the distance to offset the horizontal axis to the right of the horizontal extent.

**Related procedures:**

■ Editing Mass Haul View Styles (page 1178)

# Title Annotation Tab (Mass Haul View Style Dialog Box)

Use this tab to specify settings for the graph view and axis titles.

## Graph View Title

**Text Style**

Specifies the text style for the section view title.

**Text Height**

Specifies the current text height in drawing units.

**Title Content**

Specifies the text elements used to compose the title. Click ![icon] to open the Text Component Editor dialog box (page 1961), where you can change title format or content.

## Title Position

These controls specify the position of the mass haul view title. Use Location and Justification settings for basic positioning, then X and Y offsets for more precise placement, if necessary.

**Location**

Specifies the location of the title in relation to the mass haul view grid, either: Top, Bottom, Right, or Left.

**Justification**

Specifies the justification (alignment) of the title. If Location is Top or Bottom, justification values are Left, Center, or Right. If Location is Right or Left, justification values are Bottom, Middle, or Top.

**X Offset**

Specifies a horizontal offset from the position set by the Location and Justification values. Enter a positive number to move the title right or a negative number to move it left.

**Y Offset**

Specifies a vertical offset from the position set by Location and Justification values. Enter a positive number to move the title up or a negative number to move it down.

**Border Around The Title**

Specifies whether a border line is drawn around the title block.

**Gap**

Specifies the distance between the title and its border.

## Axis Title Text

Separate titles may be displayed at each of the axes. Use the radio buttons to select the axis title to edit. Configure title settings and click Apply. Repeat that process with each axis title.

**NOTE** The Axis Title selections do not control axis title visibility. Visibility controls for all section components are located on the Display tab.



Axis Title Text

Specifies which axis title is being controlled by the text editing tools.

Title Text

Specifies the title text for the horizontal axis. Click  to open the Text Component Editor dialog box (page 1961), where you can change title format or content.

Location

Specifies the location of the title on the axis, either: Left, Center, or Right. For more precise control, use the X and Y offsets.

Rotation

Specifies the angle for the title. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the title is not rotated.

Text Style

Specifies a text style for use in the horizontal axis title.

Text Height

Specifies the title text height (in plotted units).

X Offset

Specifies a horizontal offset from the Location setting. Enter a positive number to move the title right or a negative number to move it left.

Y Offset

Specifies a vertical offset from the Location setting. Enter a positive number to move the title up or a negative number to move it down.

**Related procedures:**

■ Editing Mass Haul View Styles (page 1178)

## Horizontal Axes Tab (Mass Haul View Style Dialog Box)

Use this tab to edit settings for the mass haul view style, such as the title format, use of tick marks, and annotations on the horizontal axes of the section view.

After you configure horizontal axis title and tick details, click Apply. Later, you can change the Select Axis setting and configure another axis on the opposite side of the grid.

Select Axis To Control

Click either Top, Middle, or Bottom to select that axis.

**Major Tick Details**

Interval

Specifies the spacing of major ticks on the horizontal axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified, either: Top, Bottom, or Center.

Text Height

Specifies the height of text used to label major ticks on the horizontal axis, using in plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. Click ![icon] to open the Text Component Editor dialog box (page 1961), where you can change text format or content.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

## Minor Tick Details

Interval

Specifies the spacing of minor ticks on the horizontal axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified, either: Top, Bottom or Center.

Text Height

Specifies the height of text used to label minor ticks on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. Click ![icon] to open the Text Component Editor dialog box, to change the text format or content.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

**Horizontal Geometry Tick Details**

Tick Size

Specifies the height of the actual tick on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified, either: Top, Bottom or Center.

Text Height

Specifies the height of text used to label minor ticks on the horizontal axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. Click ![icon] to open the Text Component Editor dialog box, to change the text format or content.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value. The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the bottom of the tick.

Y Offset

Specifies a vertical offset for the tick label from the bottom of the tick.

**Related procedures:**

■  Editing Mass Haul View Styles (page 1178)

# Vertical Axes Tab (Mass Haul View Style Dialog Box)

Use this tab to edit settings for the mass haul view style, such as the title location and format, use of tick marks, and annotations, on the vertical axis of the mass haul view.

After you configure vertical axis title and tick details, click Apply. Later, you can change the Select Axis setting and configure another axis on the opposite side of the grid.

Select Axis To Control

Click either Left or Right to select that axis.

**Major Tick Details**

Interval

Specifies the spacing of major ticks on the vertical axis. Enter a positive value in ground units.

Tick Size

Specifies the length of the actual tick on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified, either: Left, Right or Center.

Text Height

Specifies the height of text used to label major ticks on the vertical axis in plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. Click  to open the Text Component Editor dialog box (page 1961), where you can change text format or content.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the end of the tick.

Y Offset

Specifies a vertical offset for the tick label from the end of the tick.

## Minor Tick Details

Interval

Specifies the spacing of minor ticks on the vertical axis, using the actual ground units. Enter a positive value in ground units.

Tick Size

Specifies the length of the actual tick on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Justification

Specifies whether the tick is justified, either: Left, Right or Center.

Text Height

Specifies the height of text used to label minor ticks on the vertical axis, using plotted units. Enter a positive value in plotted units.

Tick Label Text

Specifies a property with which the tick is labeled. Click  to open the Text Component Editor dialog box, to change the text format or content.

Text Style

Specifies the style of the tick label text.

Rotation

Specifies the angle for the tick label. Enter a positive or negative value.The angle direction is always counterclockwise. If the value is zero (0), the label is not rotated.

X Offset

Specifies a horizontal offset for the tick label from the end of the tick.

Y Offset

Specifies a vertical offset for the tick label from the end of the tick.

**Related procedures:**

■ Editing Mass Haul View Styles (page 1178)

## Display Tab (Mass Haul View Style Dialog Box)

Use this tab to edit settings for mass haul view components and format their styles, such as visibility and color.

For more information, see Display Tab (Style Dialog Box) (page 1801).

**Related procedures:**

■ Editing Mass Haul View Styles (page 1178)

## Summary Tab (Mass Haul View Style Dialog Box)

Use this tab to edit settings for the mass haul view style.

For more information, see Summary Tab (Style Dialog Box) (page 1803).

**Related procedures:**

■ Editing Mass Haul View Styles (page 1178)

# Quantity Takeoff Criteria Dialog Box

Use this dialog box to create and edit quantity takeoff criteria.

**See also:**

■ Creating Quantity Takeoff Criteria (page 1165)

## Information Tab (Quantity Takeoff Criteria Dialog Box)

Use this tab to view or change general information for the quantity takeoff criteria.

Name

Specifies the name of the quantity takeoff criteria.

Description

Specifies the description for the quantity takeoff criteria.

**Related procedures:**

■ Creating Quantity Takeoff Criteria (page 1165)

## Material List Tab (Quantity Takeoff Criteria Dialog Box)

Use this tab to define the quantity takeoff criteria.

Add New Material

Adds a new empty material to the list. After you add the material, you can edit its type and settings and use the Define Material fields to populate it with components that define the criteria for calculating volumes.



Opens the Name Template dialog box (page 1806), where you can modify the material naming template.

Deletes the material or data component that is currently selected in the Material Name column.

**Define Material**

Data Type

Specifies the type of data that is compared and processed when defining the material. Either Surface or Corridor Shape.

Select Surface/Shape

Lists sampled surfaces and corridor shapes for this sample line group. Select a surface or corridor shape. Surface and corridor shape names in the material criteria are mapped to actual surfaces and shapes when you generate the quantity takeoff report.

**NOTE** Corridor shapes can be added only to a material with a Structure quantity type.



Adds the data specified in the Define Material fields to the selected material name.

**NOTE** If a material is not selected, the data is not added.

The properties table contains the following columns:

Material Name

Specifies the material name. Click ⊞ next to the material name to display its components (surfaces).

Condition

Specifies the condition on which to base the calculation:

- **Above**. Specifies that an area above this surface is included in the material definition. Used with Below to define two or more surfaces for cut, fill, and structures material types.

- **Below**. Specifies that an area below this surface is included in the material definition. Used with Above to define two or more surfaces for cut, fill, and structures material types.

- **Base**. Specifies that this surface is the surface to compare against the compare surface. Used with Compare to define two or more surfaces for earthworks or cut and fill material types.

- **Compare**. Specifies that this surface is the surface to compare against the base surface. Used with Base to define two or more surfaces for earthworks or cut and fill material types.

- **Include**. Specifies a corridor shape that is included in the structure type definition.

Quantity Type

Specifies the quantity type:

- **Cut**. Calculates the material to remove.

- **Fill**. Calculates the material to add.

- **Cut and Refill**. Defines an area in the section where a material is removed and refilled with fill material.

- **Earthworks**. Compares two surfaces to calculate both cut and fill areas and displays them separately.

- **Structures**. Calculates the volume of one or more corridor shapes (as defined by the shape codes that are used to define the corridor). For information about shape codes, see Understanding Point, Link, and Shape Codes (page 1406).

Shape Style

Specifies the default style used to display the material in a section view.

Cut Factor

Specifies the expansion or swell of the cut material.

Fill Factor

Specifies the contraction or shrinkage of the fill material.

Refill Factor

Specifies usability factor used to calculate how much cut material can be reused as fill.

Define From A Sample Line Group

Specifies that criteria is defined from sections in a sample line group. Opens the Define Material Criteria dialog box (page 2304).

**Related procedures:**

■ Creating Quantity Takeoff Criteria (page 1165)


# Report Quantities Dialog Box

Use this dialog box to create external reports in XML using a material list for a sample line group.

Select Alignment

Specifies the alignment. Select an alignment from the list or click  to select an alignment in the drawing. The alignments in the list are associated with at least one sample line group.

Select Sample Line Group

Specifies the sample line group to use. Select a group from the list.

Select Material List

Specifies which material list defined in the sample line group properties is used to create the report.

Select A Style Sheet

Specifies the style sheet to use for the report. Enter the path and name or click  to open the Select Style Sheet dialog box where you can browse for the style sheet.

Display XML Report

Specifies whether to display an XML report for the quantity takeoff.

**Related procedures:**

■ Analyzing Sectional Volumes (page 1168)


# Select a Sample Line Group Dialog Box

Use this dialog box to select a sample line group from which to generate quantity takeoff information.

Select Alignment

Specifies the alignment. Click  to select an alignment in the drawing. The alignments in the list are associated with at least one sample line group.

Select Sample Line Group

Specifies the sample line group to use. Select a group from the list.

■ Generating Material Lists (page 1163)

# Select Materials Dialog Box

Use this dialog box to specify the materials for the volume table.

Materials

Select or clear the check boxes to specify the addition or removal of materials from the volume table.

# Sites Dialog Boxes

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Use the following links to access information about the Sites dialog boxes.

## Site Properties Dialog Box

Use this dialog box to view or edit the general properties for a site.

**NOTE** You can set and view site parcel properties in the Site Parcel Properties (page 2015) dialog box.

**Related procedures:**

- Editing Site Properties (page 724)
- Creating a Site (page 723)

## Information Tab (Site Properties Dialog Box)

Use this tab to specify and edit the name and description for a site.

Name
Specifies the name of the site, which is used to identify the site topology collection under the Sites node in the Prospector tree.

Description
Specifies the description of the site topology collection.

**Related procedures:**

- Editing Site Properties (page 724)
- Creating a Site (page 723)

## 3D Geometry Tab (Site Properties Dialog Box)

Use this tab to define how alignments and parcel geometry should be displayed when viewed in 3D.

NOTE These settings apply only to geometry. The display properties are defined by the style that the objects reference.

**3D Geometry**

Site Display Mode

Specifies how a site's elevation is displayed in 3D:

- **Use Elevation**: Displays the site geometry at the actual elevation to which it was drawn.

- **Flatten To Elevation**: Displays the geometry flattened or projected to the elevation value specified by the Site Elevation property.

Site Elevation

Specifies the elevation used to flatten the geometry; enabled when the Site Display Mode property is set to Flatten To Elevation. All site geometry will be displayed at the specified elevation, regardless of the existing physical elevation of the geometry. The object elevations are not physically changed; they are only displayed at a single elevation.

Curve Tesselation: Mid-ordinate Distance

Specifies the mid-ordinate distance for the 3D polylines, which is used to tessellate the feature line and lot line arcs from which the polyline is being created.

**Construction Geometry**

Specifies the layers that construction geometry uses for 3D display.

When you create parcels or alignments, there may be solved and unsolved portions of the feature. The unsolved portion includes any unconnected elements of a feature.

For example, if you use the tan-tan method and create a number of lines, the whole feature is solved and the display characteristics are controlled by the parcel style. If you create some connected lines and then add a line that is not connected, the result is a piece of 'construction' geometry.

Construction Line Layer

Specifies the layer for line construction geometry. Click ⌶ to open the Layer Selection dialog box where you can select a layer.

Construction Arc Layer

Specifies the layer for arc construction geometry. Click ⌶ to open the Layer Selection dialog box where you can select a layer.

Construction Spiral Layer

Specifies the layer for spiral construction geometry. Not used.

**Related procedures:**

- Creating a Site (page 723)

- Editing Site Properties (page 724)

# Numbering Tab (Site Properties Dialog Box)

Use this tab to define how parcel and alignments components are numbered.

There are two numbering counters:

- **Automatic**. Used for numbering at creation time.

- **Manual**. Used when renumbering.

If you create a number of parcels at the same time, the Next Automatic Counter property is used to generate the starting number and the numbering automatically increments by one. For example, if the Next Automatic Counter is set to 100, the parcels would be numbered starting at 100. If you renumber the parcels, the renumbering is generated from the Next Manual Counter property. You can set the automatic and manual numbers to avoid collisions when renumbering.

### Parcels

Parcel: Next Automatic Area Counter

Specifies the starting number for new parcels to be created in the site. The numbering is incremented with creation of new parcels.

**TIP** Set this range sufficiently above the manual range to avoid collisions while renumbering manually.

Parcel: Next Manual Area Counter

Specifies the default starting number for renumbering parcels in the site. This value can also be edited in the parcel Renumbering dialog box.

**Related procedures:**

- Creating a Site (page 723)

- Editing Site Properties (page 724)

# Move To Site Dialog Box

Use this dialog box to move objects from one site to another. The contents of an entire site, including alignments, grading groups, or parcels, can be moved together.

**NOTE** If there is a naming conflict, duplicate objects must be deleted from the destination site prior to moving or copying sites or objects within a site. If duplicates objects occur, the names are automatically resolved. For example, if the destination site contains an Alignment - (1), it is renamed to Alignment - (1) (1).

Destination Site

Specifies the destination site. Select an existing site from the list, or click  to either pick a site in the drawing or create a new site, which is then returned and displayed in the list.

**NOTE** If only alignments are selected, accepting the default <None> selection places the alignments in the top-level Alignments collection in the Prospector. See Alignment and Site Interaction (page 721) for more information.

Selected Objects

Navigate in the data tree to view its subcomponents and expand the collections. The contents of the data tree are dependent on the commands entry point (that is, the location in the Prospector, or drawing, from which the command was invoked). Select or clear the check boxes to filter the objects that you want to move.

**NOTE** The check boxes have a tri-state display. If only some objects are selected under a collection, the check box is dimmed and not available.

**Related procedures:**

- Move Objects to a Site (page 726)

# Copy To Site Dialog Box

Use this dialog box to copy objects from one site to another. The contents of an entire site, including alignments, grading groups, or parcels, can be copied together.

**NOTE** If there is a naming conflict, duplicate objects must be deleted from the destination site prior to moving or copying sites or objects within a site. If duplicates objects occur, the names are automatically resolved. For example, if the destination site contains an Alignment - (1), it is renamed to Alignment - (1) (1).

Destination Site

Specifies the destination site. Select an existing site from the list, or click  to either pick a site in the drawing or create a new site, which is then returned and displayed in the list.

**NOTE** If only alignments are selected, accepting the default <None> selection places the alignments in the top-level Alignments collection in the Prospector. See Alignment and Site Interaction (page 721) for more information.

Selected Objects

Navigate in the data tree to view its subcomponents and expand the collections. The contents of the data tree are dependent on the commands entry point (that is, the location in the Prospector, or drawing, from which the command was invoked). Select or clear the check boxes to filter the objects that you want to copy.

**NOTE** The check boxes have a tri-state display. If only some objects are selected under a collection, the check box is dimmed and not available.

**Related procedures:**

■ Copy Objects to a Site (page 727)

# Surfaces Dialog Boxes

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The following topics provide information about surfaces dialog boxes.

## Edit Feature Settings - Surface Dialog Box

Use this dialog box to view and change surface-related settings.

This topic documents settings in all surface-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the drawing icon.

- Surface feature settings are listed near the top of this dialog box, after the General property group, and are identified by the surface icon.

- Surface command settings are identified by the command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### Default Styles

Use these settings to specify the default styles assigned to surfaces and surface-related labels.

Surface Default Style

Specifies the default surface style (page 692). Click in the Value column, and click ⋯ to select a style in the Surface Default Style dialog box.

Marker Style

Specifies the default marker style. Click in the Value column, and click ⋯ to select a style in the Marker Style dialog box.

Surface Spot Elevation Label Style

Specifies the default spot elevation label style. Click in the Value column, and click ⋯ to select a style in the Surface Spot Elevation Label Style dialog box.

Surface Slope Label Style

Specifies the default slope label style. Click in the Value column, and click ⋯ to select a style in the Surface Slope Label Style dialog box.

Render Material

Specifies the default render material. Click in the Value column, and click ⋯ to select a render material in the Render Material dialog box (page 1810).

For information about style selection, see Select Style Dialog Box (page 1805).

## Default Name Format

Use this setting to specify the default name format for new surfaces. Click in the Value column, and click ⋯ to make changes in the Name Template dialog box (page 1806).

## Contour Labeling Defaults

Use these settings to specify the defaults assigned to surface labels.

Display Contour Label Line

Specifies whether or not the contour label lines are displayed.

Surface Contour Label Style Major

Specifies the default major contour label style. Click in the Value column, and click ⋯ to select a style in the Surface Contour Label Style Major dialog box.

Surface Contour Label Style Minor

Specifies the default minor contour label style. Click in the Value column, and click ⋯ to select a style in the Surface Contour Label Style Minor dialog box.

Surface Contour Label Style User-defined

Specifies the default user-defined contour label style. Click in the Value column, and click ⋯ to select a style in the Surface Contour Label Style User-defined dialog box.

## Surface Defaults

Default Rebuild -Automatic

Specifies Automatic as the default setting for the Rebuild Surface command.

## Add Contour Labeling

**NOTE** This property group is displayed when accessing the settings from the AddContourLabelingGroup command.

Use this setting to specify the defaults assigned to surface labels.

Interval Along Contour

Specifies the default distance between labels along the contour. Enter a value or click 🔲 to graphically select a distance in the drawing area.

## Import Options

**NOTE** This property group is displayed when accessing the settings from the AddSurfaceFromDemFile and CreateSurfaceGridFromDemFile commands.

Use these settings to specify the defaults assigned when creating surfaces.

Use Custom Null Elevation

Specifies whether a custom null elevation is used when importing a surface from a DEM file.

Null Elevation

Specifies the default custom null elevation used when importing a surface from a DEM file.

## Surface Creation

**NOTE** This property group is displayed when accessing the settings from the CreateSurface command.

Use these settings to specify the defaults assigned when creating surfaces.

Surface Default Type

Specifies the default surface type that is used when creating a surface. For more information, see Understanding Surfaces (page 599).

Grid Surface X-Spacing

Specifies the default value for a grid surfaces X spacing.

Grid Surface Y-Spacing

Specifies the default value for a grid surfaces Y spacing.

Grid Surface Orientation

Specifies the default orientation for a grid. Enter a value or click ![icon] to graphically select an orientation in the drawing area.

## Build Options

**NOTE** This property group is displayed (in different forms) when accessing the settings from the CreateSurface and CreateSurfaceGridFromDemFile commands.

Use these settings to specify the build options assigned when creating surfaces.

**NOTE** Use these settings when creating a surface, but not when importing a surface from LandXML, DEM, or TIN.

Copy Deleted Dependent Objects

Specifies whether a drawing object is copied to the surface definition item if the object is deleted.

Exclude Elevations Less Than

Specifies whether an elevation less than a certain value should be excluded when the surface is built.

Elevation <

Specifies the elevations to exclude when the Exclude Elevations Less Than property is set to Yes. Enter a value or click ![icon] to select an elevation in the drawing area.

Exclude Elevations Greater Than

Specifies whether an elevation greater than a certain value should be excluded when the surface is built.

Elevation >

Specifies the elevations to exclude when the Exclude Elevations Greater Than property is set to Yes. Enter a value or click ![icon] to select an elevation in the drawing area.

Use Maximum Triangle Length

Specifies whether the surface triangles that exceed the length specified by the Maximum Triangle Length property are removed from the boundary of the surface (CreateSurface command only).

Maximum Triangle Length

Specifies the triangle length to use when the Use Maximum Triangle Length property is set to Yes. Enter a value or click ![icon] to select a length in the drawing area (CreateSurface command only).

Convert Proximity Breaklines to Standard

Specifies whether proximity breaklines are converted to standard breaklines when the surface is built (CreateSurface command only).

Allow Crossing Breaklines

Specifies whether breaklines can cross each other (CreateSurface command only).

Elevation To Use

Specifies the elevation to use for the crossing breaklines (CreateSurface command only):

- **Use First Breakline Elevation At Intersection**: Uses the first breakline elevation to determine the elevation at the intersection.

- **Use Last Breakline Elevation At Intersection**: Uses the last breakline elevation to determine the elevation at the intersection.

- **Use Average Elevation At Intersection**: Uses the average of the first and last breakline to determine the elevation at the intersection.

For more information on these settings, see the Build properties in Definition Tab (Surface Properties Dialog Box) (page 2346).

## Water Drop Path

**NOTE** This property group is displayed when accessing the settings from the CreateSurfaceWaterdrop command.

Use these settings to specify default water drop path options for surfaces.

Path Layer

Specifies the default layer on which to draw the water drop path. Click in the Value column, and click ![icon] to select a type of path in the Path Layer dialog box.

Path Object Type

Specifies the default type of AutoCAD object to use for the water drop path. Either 2D Polyline or 3D Polyline.

## Water Drop Marker

**NOTE** This property group is displayed when accessing the settings from the CreateSurfaceWaterdrop command.

Use these settings to specify default water drop marker options for surfaces.

Place Marker At Start Point

Specifies whether to draw a marker at the start point of the water drop path.

Start Point Marker Style

Specifies the default start point marker style. Click in the Value column, and click ![icon] to select a style in the Start Point Marker Style dialog box.

## Google Earth Options

**NOTE** This property group is displayed when accessing the settings from the ImportGEData and ImportGESurface commands.

Use these settings to specify the default number of rows and columns collected when importing Google Earth information.

Google Earth Rows

Specifies the number of rows of data that are collected when importing terrain information from Google Earth.

Google Earth Columns

Specifies the number of columns of data that are collected when importing terrain information from Google Earth.

## ⊞ SimplifySurface

**NOTE** This property group is displayed when accessing the settings from the SimplifySurface command.

Use these settings to specify default options for simplifying surfaces.

Simplify Method

Specifies the default method by which surfaces will be simplified:

- Edge Contraction: Contracts triangle edges to single points.

- Point Removal: Selects and removes surface points.

Region Options

Specifies the default method by which surfaces will be simplified:

- Use Existing Surface Border: You specify the existing surface border as the boundary of the surface simplification region.

- Specify Window/Polygon: You specify the surface simplification region within a drawn rectangle.

- Select Objects: You specify one of either 2D or 3D polylines, parcels, circles, feature lines, or survey figures.

Use Percentage Of Points To Remove

Specifies whether simplify the image by removing a percent of surface points.

Percentage Of Points To Remove

Specifies the percent of surface points to remove.

Use Maximum Change In Elevation

Specifies whether you will set a default maximum change in elevation.

Maximum Change In Elevation

Specifies the maximum allowed difference between the elevation of the original surface and the elevation of the simplified surface.

For more information about simplifying surfaces, see Simplify Surface Wizard (page 2370).

## ⊞ Export Options

**NOTE** This property group is displayed when accessing the settings from the SurfaceExportToDem command.

Use these settings to specify the defaults assigned when exporting surfaces to a DEM file.

Use Custom Null Elevation

Specifies whether a custom null elevation is used when exporting to a DEM file.

Null Elevation

Specifies the default custom null elevation used when exporting to a DEM file.

Grid Spacing

Specifies the spacing between grids when exporting to a DEM file.

### 🖳 Add Data Options

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**NOTE** This property group is displayed when accessing the settings from the AddSurfaceContours, AddSurfaceBoundaries, and AddSurfaceBreaklines commands.

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Use these settings to specify the defaults assigned when adding contours, boundaries, and breaklines to a surface.

Default Weeding Distance

Specifies the default weeding distance.

Default Weeding Angle

Specifies the default weeding angle.

Default Supplementing Distance

Specifies the default supplementing distance.

Default Mid-ordinate Distance

Specifies the default mid-ordinate distance.

Default Boundary Type

Specifies the default boundary type.

Default Non-destructive Setting

Specifies the default selection of a type of breakline (destructive of non-destructive).

**Related procedures:**

- Editing Surface Settings (page 687)
- Adding and Editing Surface Data (page 611)
- Analyzing Surfaces (page 708)

# Surface Style Dialog Box

Use this dialog box to define a surface style, which controls the way a surface and its components, such as watersheds and contours, are displayed.

**See also:**

- Surface Styles and Visualization (page 692)
- Analyzing Surfaces (page 708)

## Information Tab (Surface Style Dialog Box)

Use this tab to change the surface style name and description information, and to review details, such as when you recently modified the style.

Name

Specifies a name for the current style. The style name identifies the style on the Settings tab in Toolspace and is listed in the Styles dialog box. Changes to a style name are reflected in the title bar of the Surface Style dialog box.

Description

Specifies a description for the current style.

Created By

Displays the AutoCAD login name of the person who created the style.

Date Created

Displays the style creation date and time.

Last Modified By

Displays the AutoCAD login name of the person who last modified the style.

Date Modified

Displays the last date and time of the style modifications.

**Related procedures:**

- Surface Styles and Visualization (page 692)

- Analyzing Surfaces (page 708)

# Borders Tab (Surface Style Dialog Box)

Use this tab to specify settings for the border component of a surface object.

Borders display the actual extents of the surface, which can be different from boundaries (page 612), which are used to define the surface extents.

## 3D Geometry

Border Display Mode

Specifies how to display the border:

- **Use Surface Elevation**: Displays the border at its actual elevation values.

- **Flatten Elevations**: Flattens or projects the border to the elevation value specified by the Flatten Borders To Elevation property.

- **Exaggerate Elevation**: Scales the border by the factor specified in the Exaggerate Borders By Scale Factor property.

Flatten Borders To Elevation

Specifies the elevation used to flatten the borders; available only when Border Display Mode is set to Flatten Elevations.

Exaggerate Borders By Scale Factor

Specifies the scale factor used to exaggerate the borders; available when Border Display Mode is set to Exaggerate Elevation.

## Border Types

Display Exterior Borders

Specifies whether to display outer borders. The outer border is the outer extents of the surface.

Display Interior Borders

Specifies whether to display all interior borders. This option is useful when you want to build footprints or "holes" in your surface where survey data is not present or applicable, and you do not want to represent elevations for that area.

**Datum**

Use Datum

Specifies whether or not the outer boundary can display a datum. The datum is displayed at the elevation specified in the Datum Elevation property.

Project Grid To Datum

Specifies whether the outer boundary segments are projected to the datum. The projection of outer boundary segments to the datum is only visible when viewing the surface object in 3D.

Datum Elevation

Specifies the datum elevation. The datum is only visible when the border is displayed in 3D.

**Related procedures:**

■ Surface Styles and Visualization (page 692)

# Contours Tab (Surface Style Dialog Box)

Use this tab to specify settings for the contour components of the surface object.

## 3D Geometry

Contour Display Mode

Specifies the display properties of the contours:

■ **Use Surface Elevation**: Displays the contours at their actual elevation values.

■ **Flatten Elevations**: Flattens or projects the contours to the elevation value specified by the Flatten Contours To Elevation property.

■ **Exaggerate Elevation**: Scales the contours by the factor specified in the Exaggerate Contours By Scale Factor property.

Flatten Contours To Elevation

Specifies the elevation used to flatten the contours; available only when the Contour Display Mode is set to Flatten Elevations.

Exaggerate Contours By Scale Factor

Specifies the scale factor used to exaggerate the contours; available only when the Contour Display Mode is set to Exaggerate Elevation.

## Legend

Contour Legend Style

Specifies the default contour table (legend) style. Click  to open the Contour Legend Style dialog box where you can select a style.

## Contour Intervals

Base Elevation

Specifies the relative base elevation for the contour intervals. The contour at the base elevation is a major contour.

Minor Interval

Specifies the distance between minor contours. This value is displayed in the drawings linear units. The interval is relative to the Base Elevation value.

Major Interval

Specifies the distance between major contours. This value is displayed in the drawings linear units. The value entered here is divisible by the minor interval. The interval is relative to the base elevation value.

## Contour Ranges

Specifies the default values for the contour analysis type. The contour analysis is created using the Surface Properties - Analysis tab (page 2349).

Group Values By

Specifies the creation options for the contour ranges:

- **Equal Interval**: Divides the data into the specified number of ranges from the minimum value to the maximum. Also known as equal step, this method often over-generalizes the data, with large sets of values in one group while others have a sparse set of values.

- **Quantile**: Divides the data so that the specified number of ranges contains an equal number of values. Also known as equal count, this method is most appropriate where the data values are linear (equally distributed).

- **Standard Deviation**: Calculates and divides the data based on how far data values differ from the arithmetic mean. This method is most effective when the data approximates a normal distribution (bell-shaped curve), and because of this curve preference, it is best used with an even number of ranges. Standard deviations are often used to highlight how far above or below a specific value is in relation to the mean value.

Number Of Ranges

Specifies the number of ranges to be used. For information about setting the display properties for contours within each range, see the Major Display and Minor Display columns in the display table on this tab.

Range Precision

Specifies the formatting and rounding of range values. The rounding only affects the range value calculations and resulting range intervals. It does not affect the actual data values.

Use Color Scheme

Specifies whether or not major and minor color schemes are active.

Major Color Scheme

Specifies the major color scheme for the contour range. Available only if you set Use Color Scheme to True.

Minor Color Scheme

Specifies the minor color scheme for the contour range. Available only if you set Use Color Scheme to True.

## Contour Depressions

Display Depression Contours

Specifies whether or not depression contours are displayed.

Tick Mark Interval

Specifies the spacing of the tick marks along the depression contour.

Tick Mark Length

Specifies the length of the tick mark along the depression contour. The tick mark is drawn on the downhill side of the depression contour.

## Contour Smoothing

**NOTE** Do not enable contour smoothing if you plan to use surface smoothing. For more information, see Smoothing Surfaces (page 662).

Smooth Contours

Specifies whether or not to use contour smoothing. Click True to smooth contours in accordance with the Smoothing Type parameter and the value set in the Contour Smoothing slider.

**NOTE** Use the Contour Smoothing slider (at the bottom of the dialog box) to increase or decrease the smoothing.

Smoothing Type

Specifies the type of smoothing used to display contour curves:

- **Add Vertices**: Adds vertices along the contours. This option supplements points on the contours when smoothing them, giving them a more noticeably curved appearance, while maintaining the highest level of integrity relative to the surface.

- **Spline Curve**: Passes a spline curve through the contour points. This option creates the smoothest contour representation.

### Display table

Use the following settings for each contour range specified using the Contour Ranges property group in the Properties list:

Number

Displays the number for the contour range. The numbers match the value set in the Number of Ranges property.

Major Display

Specifies a display properties for major contours. Click ▦ to select a linetype. Click ▤ to select a lineweight. Click ▥ to select a color.

Minor Display

Specifies a display properties for minor contours. Click ▦ to select a linetype. Click ▤ to select a lineweight. Click ▥ to select a color.

**NOTE** After you specify the Contour Ranges properties and set the display properties, run the contour analysis to see the changes in the surface. For information, see Creating Surface Analysis (page 708).

### Contour Smoothing slidebar

Use the Contour Smoothing slidebar to give the contours a smoother appearance. The slidebar is enabled if the Smooth Contours property (under the Contour Smoothing property group) is set to True.

**Related procedures:**

- Surface Styles and Visualization (page 692)

- Contours (page 629)

# Grid Tab (Surface Style Dialog Box)

Use this tab to specify the settings for the grid line components of the surface object.

### 3D Geometry

Grid Display Mode

Specifies how the grid is displayed:

- **Use Surface Elevation**: Displays the grid at its actual elevation values.

- **Flatten Elevations**: Flattens or projects the grid to the elevation value specified by the Flatten Grid To Elevation property.

- **Exaggerate Elevation**: Scales the grid by the factor specified in the Exaggerate Grid By Scale Factor property.

Flatten Grid To Elevation

Specifies the elevation used to flatten the grid; available only when you set the Grid Display Mode to Flatten Elevations.

Exaggerate Grid By Scale Factor

Specifies the scale factor used to exaggerate the grid; available only when you set the Grid Display Mode to Exaggerate Elevation.

### Primary Grid

Specifies whether the primary grid lines are used in the grid display as well as the primary grid interval value and orientation.

Use Primary Grid

Specifies whether or not the primary grid lines are used in the grid display. Click True to use the primary grid lines in the grid display.

Interval

Specifies the primary grid interval value in drawing linear units. The primary grid lines are the north-south, or longitudinal grid lines.

Orientation

Specifies the angle of orientation for the primary grid lines. The angle is clockwise (+), or counterclockwise (-) from the North-South meridian.

### Secondary Grid

Specifies whether the secondary grid lines are used in the grid display as well as the secondary grid interval value and orientation.

Use Secondary Grid

Specifies whether or not the secondary grid lines are used in the grid display. Click True to use the secondary grid lines in the grid display.

Interval

Specifies the secondary grid interval value in drawing linear units. The secondary grid lines are the east-west, or latitudinal, grid lines.

Orientation

Specifies the angle of orientation for the secondary grid lines. The angle is clockwise (+), or counterclockwise (-) from the East-West meridian.

**Related procedures:**

- Surface Styles and Visualization (page 692)


# Points Tab (Surface Style Dialog Box)

Use this tab to specify the settings for point components of the surface objects.

**3D Geometry**

Point Display Mode

Specifies the points display properties:

- **Use Surface Elevation**: Displays the points at their actual elevation values.

- **Flatten Elevations**: Flattens or projects the points to the elevation value specified by the Flatten Points To Elevation property.

- **Exaggerate Elevation**: Scales the points by the factor specified in the Exaggerate Points By Scale Factor property.

Flatten Points To Elevation

Specifies the elevation used to flatten the points; available only when you set the Point Display Mode to Flatten Elevations.

Exaggerate Points By Scale Factor

Specifies the scale factor used to exaggerate the points; available only when you set the Point Display Mode to Exaggerate Elevation.

## Point Size

Point Scaling Method

Specifies the scaling method, which, in combination with the Point Units value, determines the point symbol size:

- **Use Drawing Scale**: Specifies that the point size is a product of the value specified in the Point Units property and the drawing scale. For example, with a 1"=40' drawing scale, using a Point Units value of 0.1 is the equivalent of using Size In Absolute Units with a value of 4'.

- **Size Relative To Screen**: Specifies that the point symbol size is a percentage of the screen size. The percentage value is specified by the Point Units property.

- **Size In Absolute Units**: The value specified in the Point Units property is the absolute point symbol size.

Point Units

Specifies the value used for the point size.

## Point Display

Data Point Symbol

Specifies the symbology of the data point display. Click  to select a data point symbol.

Data Point Color

Specifies the color of the data point. Click  to select a data point color.

Derived Point Symbol

Specifies the symbology of the derived point display. A derived point is not a data point, but a calculated one (for example, from a surface smoothing operation). Click  to select a derived point symbol.

Derived Point Color

Specifies the color of the derived point. Click  to select a derived point color.

Non-Destructive Point Symbol

Specifies the symbology of the non-destructive point display. Non-destructive points are creating when adding non-destructive breaklines. Click  to select a non-destructive point symbol.

Non-Destructive Point Color

Specifies the color of the non-destructive point. Click ▣ to select a non-destructive point color.

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**NOTE** If you specify a color for the points using the Point Display properties, it overrides the color specified on the Display tab. The default color value is set to ByBlock when a new style is initialized. When set to ByBlock, the color specified on the Display tab is used.

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**Related procedures:**

■ Surface Styles and Visualization (page 692)

# Triangles Tab (Surface Style Dialog Box)

Use this tab to specify the properties of the triangle components of the surface object.

### 3D Geometry

Triangle Display Mode

Specifies the triangles display properties:

■ **Use Surface Elevation**: Displays the triangles at their actual elevation values.

■ **Flatten Elevations**: Flattens or projects the triangles to the elevation value specified by the Flatten Triangles To Elevation property.

■ **Exaggerate Elevation**: Scales the triangles by the factor specified in the Exaggerate Triangles By Scale Factor property.

Flatten Triangles To Elevation

Specifies the elevation used to flatten the triangles; available only when you set the Triangle Display Mode to Flatten Elevations.

Exaggerate Triangles By Scale Factor

Specifies the scale factor used to exaggerate the triangles; available only when you set the Triangle Display Mode to Exaggerate Elevation.

**Related procedures:**

■ Surface Styles and Visualization (page 692)

# Watersheds Tab (Surface Style Dialog Box)

Use this tab to control the display of watersheds.

### 3D Geometry

Watershed Display Mode

Specifies the watersheds display properties:

■ **Use Surface Elevation**: Displays the watersheds at their actual elevation values.

■ **Flatten Elevations**: Flattens or projects the watersheds to the elevation value specified by the Flatten Watersheds To Elevation property.

■ **Exaggerate Elevation**: Scales the watersheds by the factor specified in the Exaggerate Watersheds By Scale Factor property.

**Flatten Watersheds To Elevation**

Specifies the elevation used to flatten the watersheds; available only when you set the Watershed Display Mode to Flatten Elevations.

**Exaggerate Watersheds By Scale Factor**

Specifies the scale factor used to exaggerate the watersheds; available only when you set the Watershed Display Mode to Exaggerate Elevation.

## Point Size

Specifies the size and scaling method of the drain target points.

**Watershed Scaling Method**

Specifies the scaling method, which, in combination with the Watershed Units value, determines the point symbol size:

- **Use Drawing Scale**: Specifies that the point size is a product of the value specified in the Watershed Units property and the drawing scale. For example, with a 1"=40' drawing scale, using a Watershed Units value of 0.1 is the equivalent of using Size In Absolute Units with a value of 4'.

- **Size Relative To Screen**: Specifies that the point symbol size is a percentage of the screen size. The percentage value is specified by the Watershed Units property.

- **Size In Absolute Units**: The value specified in the Watershed Units property is the absolute point symbol size.

**Watershed Units**

Specifies the value for the point size.

## Surface

**Surface Watershed Label Style**

Specifies the default watershed label style. Click ⬚ to select a style.

## Legend

**Watershed Legend Style**

Specifies the default watershed table (legend) style. Click ⬚ to select a style.

## Boundary Point, Boundary Segment, Depression, Flat Area, Multi-drain, Multi-drain Notch

The above watershed categories have the following common properties.

**Color**

Specifies the color of the watershed. Click ⬚ to select the color.

**Linetype**

Specifies a linetype for the watershed. Click ⬚ to select a linetype.

**Use Hatching**

Specifies whether or not to use a hatch pattern for the watershed.

**Hatch Pattern**

Specifies the hatch pattern to use for the watershed; available only when you set the Use Hatching parameter mode to True. Click ⬚ to open the Hatch Properties Dialog Box (page 2365) where you can select a hatch pattern.

**Draw Drain Target Point**

Specifies whether or not to display drain target points for the watershed.

Drain Target Point Display

Specifies the symbology of the drain target point display. Click ⬚ to select a drain target point symbol.

Drain Target Point Color

Specifies the color of the drain target point. Click ⬚ to select a drain target point color.

Draw Drain Target Segment

Specifies whether or not to display drain target segments for the watershed.

Drain Target Segment Color

Specifies the color of the drain target segment. Click ⬚ to select a drain target segment color.

Drain Target Segment Linetype

Specifies a linetype for the drain target segment. Click ⬚ to select a linetype.

**Related procedures:**

■  Watersheds (page 675)

■  Surface Styles and Visualization (page 692)

# Analysis Tab (Surface Style Dialog Box)

Use this tab to specify the display parameters for the surface analysis displays that have common view properties.

## Directions

Specifies the display properties for directional (aspect) analysis. Renders surface triangles according to the direction that they face.

Group By

Specifies the direction ranges creation properties:

■  **Equal Interval**: Divides the data into the specified number of ranges from the minimum value to the maximum. Also known as equal step, this method often over-generalizes the data, with large sets of values in one group while others have a sparse set of values.

■  **Quantile**: Divides the data such that the specified number of ranges contains an equal number of values. Also known as equal count, this method is most appropriate where the data values are linear (equally distributed).

■  **Standard Deviation**: Calculates and divides the data based on how far data values differ from the arithmetic mean. This method is most effective when the data approximates a normal distribution (bell-shaped curve), and because of this curve preference, it is best used with an even number of ranges. Standard deviations are often used to highlight how far above or below a specific value is in relation to the mean value.

Number of Ranges

Specifies the number of ranges to be used.

Range Precision

Specifies the range precision value.

Display Type

Specifies the entity type to be drawn within the analysis view:

- **3D Faces**: Generates a surface analysis display containing 3D Faces. Useful when viewing the object in 3D. If you use the AutoCAD Explode command, the analysis view explodes to 3D Face entities.

- **2D Solid**: Generates a surface analysis display containing 2D Solid hatching. Useful when viewing the object in 2D. If you use the AutoCAD Explode command, the analysis view explodes to Solid entities.

- **Hatch Solid**: Generates hatching within polygons containing triangle faces that match the criteria for a specific range. Useful when viewing the object in 2D. If you use the AutoCAD Explode command, the analysis view explodes to Hatch entities.

- **Mesh**: Generates a surface analysis display containing a mesh. Useful when viewing the object in 3D.

Scheme

Specifies the color scheme for the range.

Legend Style

Specifies the default analysis table (legend) style. Click ... to select a style.

Directions Display Mode

Specifies watersheds display properties:

- **Use Surface Elevation**: Displays the directions at their actual elevation values.

- **Flatten Elevations**: Flattens or projects the directions to the elevation value specified by the Flatten Directions To Elevation property.

- **Exaggerate Elevation**: Scales the directions by the factor specified in the Exaggerate Directions By Scale Factor property.

Flatten Directions To Elevation

Specifies the elevation used to flatten the directions display. Available only when you set the Directions Display Mode to Flatten Elevations.

Exaggerate Directions By Scale Factor

Specifies the scale factor used to exaggerate the directions display. Available only when you set the Directions Display Mode to Exaggerate Elevation.

## Elevations

Specifies the display properties for elevation banding analysis. Renders surface triangles within an elevation range.

**NOTE** Elevations analysis display properties are the same as the Directions properties.

## Slopes

Specifies the display properties for slope analysis. Renders surface triangles within a slope range.

**NOTE** Slopes analysis display properties are the same as the Directions properties.

## Slope Arrows

Used for slope direction analysis. Places a slope directional arrow at each triangle centroid. Arrow color is based on the color assigned to a slope range (similar to slope analysis).

Uses the same display properties as the Directions properties, with the following additional properties:

Arrow Type

Specifies the type of arrow to use:

- **Filled:** 

- **Closed:** 

- **Open:** 

- **Double:** 

Arrow Length

Specifies the length of the slope arrow that is displayed in a Slope Arrows surface analysis view.

**Related procedures:**

- Analyzing Surfaces (page 708)

## Display Tab (Surface Style Dialog Box)

Use this tab to control the visibility and display of the components that make up a surface. You can change the visibility, color, or layer for the surface components at various phases in a project.

Surface components include:

- **Points:** All surface points for the TIN or grid surfaces.

- **Triangles**: TIN face lines.

- **Border**: Interior and exterior border lines.

- **Major Contour**: Contour lines defined as major interval.

- **Minor Contour**: Contour lines defined as minor interval.

- **User Contours**: User-defined contour lines.

- **Gridded**: Primary and secondary grid lines.

- **Directions**: Direction or aspect analysis display.

- **Elevations**: Elevation banding analysis display.

- **Slopes**: Slope analysis display.

- **Slope Arrows**: Slope arrow analysis display.

- **Watersheds**: Watershed analysis display.

**NOTE** Surface styles have unique display values and varying numbers of displayed components, depending on whether you select 2D or 3D view.

For more information about the properties in this tab, see Display Tab (Style Dialog Box) (page 1801).

## Summary Tab (Surface Style Dialog Box)

Use this tab to view all the surface style properties. For more information about the properties in this tab, see Summary Tab (Style Dialog Box) (page 1803).

# Surface Properties Dialog Box

Use this dialog box to change the properties of a surface.

Surface properties control which data is included in the surface. You also use the Surface Properties dialog box to perform analysis calculations on the surface.

**See also:**

■ Understanding the Surface Definition (page 611)

■ Excluding Data from the Surface Build (page 667)

■ Analyzing Surfaces (page 708)

## Information Tab (Surface Properties Dialog Box)

Use this tab to view or change general information for the surface.

Name

Specifies the name of the current surface.

Description

Specifies the description of the current surface.

Object Style

Specifies the default surface style used to display a surface. Select a style in the list or use the standard selection tools. For more information about the standard selection tools, see Select Style Dialog Box (page 1805).

Render Material

Specifies the default render material used to display a surface. Select a render material in the list or use the standard selection tools. For more information, see Applying Render Materials to Objects (page 1578), Select Render Material Dialog Box (page 2366).

Object Locked

Specifies whether the surface is locked and cannot be edited.

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (does not affect tooltips over toolbar icons).

**Related procedures:**

■ Creating Surfaces (page 601)

■ Project Management (page 125)

## Definition Tab (Surface Properties Dialog Box)

Use this tab to switch various data and edit items on or off, or remove surface definition items.

The bottom part of the tab lists the surface definition (operation types) in the order that the operations were performed. You can reorder the surface operations in the list.

**NOTE** Properties that are not applicable for the current surface type are unavailable.

## Build

NOTE You can set the default settings for the Build properties in the Edit Command Settings - CreateSurface dialog box, which is accessed from the Settings tab on the Toolspace (under <drawing-name> ➤ Surface ➤ Commands, right-click ▣ CreateSurface and click Edit Commands Settings).

Copy Deleted Dependent Objects

Specifies whether a drawing object is copied to the surface definition item if the object is deleted:

- **Yes**: Copies the drawing object data to the surface definition if you delete the drawing object data. When you add a surface boundary, breakline, or point group to a surface, the surface definition stores the Object ID. If the drawing object, such as the original polyline used to define the surface boundary, is deleted from the drawing, the coordinates of the polyline vertices are copied into the surface definition and saved. Thus, the surface boundary definition remains in the surface definition.

- **No**: Deletes the surface boundary, breakline, or point group definition if the drawing object is deleted. The definition item is removed from the surface definition when the surface is rebuilt.

Exclude Elevations Less Than

Specifies whether an elevation less than a certain value should be excluded when the surface is built. Click Yes to exclude elevations less than the value specified in the Elevation < when the surface is built.

Elevation <

Specifies the elevations to exclude when the Exclude Elevations Less Than property is set to Yes.

Exclude Elevations Greater Than

Specifies whether an elevation greater than a certain value should be excluded when the surface is built. Click Yes to exclude elevations greater than the value specified in the Elevation > when the surface is built.

Elevation >

Specifies the elevations to exclude when the Exclude Elevations Greater Than property is set to Yes.

Use Maximum Triangle Length

Specifies whether the surface triangles that exceed the length specified by the Maximum Triangle Length property are removed from the boundary of the surface. Click Yes to remove triangles with a length greater than the value specified in the Maximum Triangle Length when the surface is built.

Maximum Triangle Length

Specifies the triangle length to use when the Use Maximum Triangle Length property is set to Yes.

Convert Proximity Breaklines to Standard

Specifies whether proximity breaklines are converted to standard breaklines when the surface is built. For more information, see Types of Breaklines (page 619). Click Yes to convert proximity breaklines to standard breaklines.

Allow Crossing Breaklines

Specifies whether breaklines can cross each other. Click Yes to enable the correction of crossing breaklines when they occur. If a new breakline touches an existing breakline, you can specify what elevation to use at the calculated point of intersection. The XY coordinate location is calculated and the elevation is set by the Elevation To Use property.
This setting does not apply to breakline intersections that occur at the endpoints of breaklines. You can change the elevation of such points using the surface Modify Point operation. For more information, see Modifying Surface Points (page 654).

Elevation To Use

Specifies the elevation to use for the crossing breaklines:

- **Use First Breakline Elevation At Intersection**: Uses the first breakline elevation to determine the elevation at the intersection.

- **Use Last Breakline Elevation At Intersection**: Uses the last breakline elevation to determine the elevation at the intersection.

- **Use Average Elevation At Intersection**: Uses the average of the first and last breakline to determine the elevation at the intersection.

## Data Operations

All Data Operations properties specify whether a data definition of a certain type is included in the surface build.

- **Yes**: All data operations of the specified type are selected in the Operation Type list and included in the surface build.

- **No**: All data operations of the specified type are cleared in the Operation Type list and not included in the surface build.

If the Operation Type list contains mixed selected and cleared items of a specific data type, the specific data operation is displayed as **Varies**.

For information about data operations supported by each surface type, see Understanding the Surface Definition (page 611).

## Edit Operations

All Edit Operations specify whether surface operations of a specific type are included in the surface build.

- **Yes**: All edit operations of specified type are selected in the Operation Type list and included in the surface build.

- **No**: All edit operations of the specified type are cleared in the Operation Type list and not included in the surface build.

If the Operation Type list contains mixed selected and cleared items of a specific edit type, the specific edit is displayed as **Varies**.

For information on edit operations supported by each surface type, see Surface Editing Operations (page 650).

## Operation Type

The Operation Type list displays the surface operations and their applicable parameters in the order in which they were performed. Clear an operation to remove it from the surface definition.

Edit the definition list by selecting or clearing a definition item, or deleting the item from the list. When definition items are cleared, the surface is updated, but the surface still has a reference to the definition item. If the definition item is deleted from the list, the definition item data is permanently removed from the surface.

Reorder surface operations in the surface definition using the following controls:

⇧

Moves a selected operation to the top of the list.

⇧

Moves a selected operation one position up.

Moves a selected operation one position down.



Moves a selected operation to the bottom of the list.

For information about errors that can occur in the Operation Type list, see Surface Errors and Issues (page 647).

**Related procedures:**

- Understanding the Surface Definition (page 611)

- Excluding Data from the Surface Build (page 667)

- Editing and Viewing the Surface Definition (page 681)

# Analysis Tab (Surface Properties Dialog Box)

Use this tab to modify the properties of a selected surface analysis type.

**NOTE** The Analysis tab displays varying fields and options depending on the type of analysis that you select.

Analysis Type

Specifies the current analysis type properties that you can modify.

**NOTE** You set the default values for the analysis in the Analysis tab of the Surface Style dialog box.

- **Contours**: Specifies the contours analysis properties.

- **Directions**: Specifies the directions analysis properties.

- **Elevations**: Specifies the elevations analysis properties.

- **Slopes**: Specifies the slopes analysis properties.

- **Slope Arrows**: Specifies the slope arrows analysis properties.

- **User-Defined Contours**: Specifies the user-defined contours analysis properties.

- **Watersheds**: Specifies the watersheds analysis properties.

Preview

Displays a preview of the legend table for the current analysis type. Clear the check box to disable the preview option.

Legend

Specifies the legend style for the current analysis type.

**NOTE** You set the default style on the Analysis tab of the Surface Styles dialog box.

## Ranges

Specifies the number of ranges to use for contours, directions, elevations, slopes, slope arrows, and user-defined contour analysis.

Number

Specifies the numbers of ranges in the analysis. Enter a number or click the up/down arrows to increment the value.

Generates the analysis and updates the Range Details group box with the analysis information.

## Watershed Parameters

Displayed if Analysis Type is set to Watersheds. Specifies the minimum depression depth and area for watershed analysis.

Minimum Average Depth Threshold

Specifies the minimum average depth at which a depression in the surface may be considered a watershed. This prevents minor depression depths from being defined as watersheds.

Merge Adjacent Boundary Watersheds

Specifies that adjacent boundary segment and or boundary point watersheds (along the edges of the outer surface border) should be merged.

Generates the analysis and updates the Details group box with the analysis information.

## Range Details

**NOTE** The Range Details area of the Analysis tab displays varying fields and columns depending on the type of analysis that you have selected.

Scale Scheme To Fit

Specifies whether the number of entries is scaled or read sequentially:

■ Selected: The number of entries in the scheme is scaled to uniformly cover the number of ranges, that is, an even sampling is taken from the scheme between the first and last entries.

■ Cleared: The scheme is read sequentially starting at the first entry.

**NOTE** If there are fewer entries in the scheme than the number of values in the list of ranges, AutoCAD Civil functions as if the check box is cleared. This means that the scheme is read sequentially until all the entries in the scheme have been used. In all cases, the last entry in the scheme is used when there are more ranges than scheme entries.

## Range Details

Specifies various details in table form depending on the type of analysis that is performed. Details are displayed in the following columns:

ID

Displays an index number for each range.

Description

Specifies a description for user-defined contours and watersheds.

Elevation

Displays the elevation value of the user-defined contour.

Minimum Elevation/Direction/Slope

Displays the minimum range value for the analysis view.

Maximum Elevation/Direction/Slope

Displays the maximum range value for the analysis view.

Scheme

Specifies a color preview assigned by the Scheme property in the surface styles Analysis tab (page 2343). Double-click the column to select a different color.

Major Contour

Specifies how the ranges major contours are displayed:

- Click ⬛ to select a display color for the major contour display.

- Click ▤ to select a display lineweight for the major contour display.

- Click ▦ to select a display linetype for the major contours.

**NOTE** You set the default display for the surface style on the Analysis tab (page 2343).

Minor Contour

Specifies how the ranges minor contours are displayed:

- Click ⬛ to select a display color for the minor contour display.

- Click ▤ to select a display lineweight for the minor contour display.

- Click ▦ to select a display linetype for the minor contours.

**NOTE** You set the default display for the surface style on the Analysis tab (page 2343).

Type

Displays the watershed type and specifies if the individual watershed is displayed or hidden in the drawing. Click the light bulb icon to display 💡 or hide 💡 .

**NOTE** To change the display for the watersheds by type rather than individually, click 💡 above the Details table.

Drains Into

Displays a comma-delimited string containing the subarea watershed IDs that the subarea watershed drains into. For example, the string "4,5,6" indicates that the subarea drains into subareas 4, 5, and 6.

Segment Display

Specifies how the segments of the watershed subarea are displayed:

- Click ⬛ to select a display color for the segment.

- Click ▤ to select a display lineweight for the segment.

- Click ▦ to select a display linetype for the segment.

**NOTE** You set the default display for the surface style on the Analysis tab (page 2343).

Area Display

Specifies and previews the hatch pattern for the watershed subarea.

- Click the hatch pattern preview to change the pattern using the Hatch Properties (page 2365) dialog box.

- Click ⬛ to select a display color for the hatching.

**NOTE** You set the default display for the surface style on the Analysis tab (page 2343).

💡

Click to open the Watershed Display (page 2359) dialog box where you can hide or display watersheds by type.

**Related procedures:**

■ Analyzing Surfaces (page 708)

■ Watersheds (page 675)

# Statistics Tab (Surface Properties Dialog Box)

Use this tab to display surface statistics that are based on the current state of the surface.

## General

Displays the general statistics for a surface:

Revision Number

Displays the revision number of the surface.

Number of Points

Displays the number of vertex points in the surface.

Minimum X Coordinate

Displays the minimum X coordinate for the surface area.

Minimum Y Coordinate

Displays the minimum Y coordinate for the surface area.

Maximum X Coordinate

Displays the maximum X coordinate for the surface area.

Maximum Y Coordinate

Displays the maximum Y coordinate for the surface area.

Minimum Elevation

Displays the minimum elevation value found in the surface.

Maximum Elevation

Displays the maximum elevation value found in the surface.

Mean Elevation

Displays the mean or average elevation of all the points on the surface.

## Extended

2D Surface Area

Displays the two-dimensional area of the surface.

3D Surface Area

Displays the three-dimensional area of the surface.

Minimum Grade/Slope

Displays the minimum grade or slope found on the surface. Expressed in the units specified in the Edit Feature Settings - Surface Dialog Box (page 2329).

Maximum Grade/Slope

Displays the maximum grade or slope found on the surface. Expressed in the units specified in the Edit Feature Settings - Surface Dialog Box (page 2329).

Mean Grade/Slope

The mean or average grade or slope for the surface. Expressed in the units specified in the Edit Feature Settings - Surface Dialog Box (page 2329).

## TIN

**NOTE** The following properties are displayed for TIN surfaces.

Number Of Triangles

Displays the number of triangles found in the surface.

Maximum Triangle Area

Displays the area of the largest triangle contained in the surface.

Minimum Triangle Area

Displays the area of the smallest triangle contained in the surface.

## Grid

**NOTE** The following properties are displayed for grid surfaces.

X Spacing

Displays the distance between the grid lines in the X direction.

Y Spacing

Displays the distance between the grid lines in the Y direction.

Orientation

Displays the orientation of the grid (in degrees). The angle is clockwise (+), or counterclockwise (-) from the North-South meridian.

## Volume

**NOTE** The following properties are displayed for volume surfaces.

Base Surface

Displays the name of the base surface from which the volume surface is generated.

Comparison Surface

Displays the name of the comparison surface from which the volume surface is generated.

Cut volume (unadjusted)

Displays the total cut volume for the surface, without a cut factor applied.

Fill volume (unadjusted)

The total fill volume for the surface, without a fill factor applied.

Net volume (unadjusted)

The difference between the cut and fill volumes.

**Related procedures:**

■ Viewing Surface Statistics (page 689)

# Create Surface Dialog Box

Use this dialog box to specify the initial surface creation parameters, including surface type and layer.

Type

Specifies the type of surface to create.

Surface Layer

Displays the layer on which the surface is created.



Click to open the Object Layer dialog box where you can select a different layer for the surface.

The properties table section of the dialog box displays varying properties depending on the type of surface selected in the Type field.

## Information

Name

Specifies the name of the surface.

> **NOTE** To name the surface, click its default name and enter a new name, or click the Name Template button and use the name template. For more information, see Name Template Dialog Box (page 1806).

Description

Specifies a description of the surface.

Style

Specifies the style for the surface. Click ⋯ to open the Select Surface Style (page 1801) dialog box where you can select a different style.

Render Material

Specifies the render material (page 1577). Click ⋯ to select a render material.

## Grid Parameters

This property group is displayed when Grid Surface is selected in the Type list.

Grid-X Spacing

Specifies the X distance between the grid lines. When selected, you can click ⬚ to digitize the spacing in the drawing area.

Grid-Y Spacing

Specifies the Y distance between the grid lines. When selected, you can click ⬚ to digitize the spacing in the drawing area.

Orientation

Specifies the direction for the grid in the X and Y directions. When selected, you can click ⬚ to pick two points in the drawing to define the orientation direction.

## Volume Surfaces

Base Surface

Specifies the base surface from which the volume surface is generated. Enter the surface name or click ⋯ to open the Select Base Surface (page 1810) dialog box where you can select the surface from the list.

Comparison Surface

Specifies the comparison surface from which the volume surface is generated. Enter the surface name or click ⬚ o open the Select Comparison Surface (page 1810) dialog box where you can select the surface from the list.

Cut Factor

Specifies the expansion value of the material (page 1166).

Fill Factor

Specifies the compaction value of the material (page 1166).

**Related procedures:**

■  Creating Surfaces (page 601)

# Add Contour Data Dialog Box

Use this dialog box to add contour data to the surface definition of the selected surface. Weeding and supplementing factors are used to add or remove vertices along a contour.

Description

Specifies the description of the contour data operation. If no description is specified, a default naming convention is used: "Contour Data<#>". For example, if you do not enter a description, the first contour data operation is named "Contour Data1," the second contour is named "Contour Data2," and so on.

## Weeding Factors

Distance

Specifies the contour weeding distance. Enter the distance value or click ⬛ to select it in the drawing.

Angle

Specifies the contour weeding angle. Enter the angle value or click ⬛ to select it in the drawing.

## Supplementing Factors

Distance

Specifies the contour supplementing distance. Enter the distance value or click ⬛ to select it in the drawing.

Mid-Ordinate Distance

Specifies the contour mid-ordinate distance. Enter the distance value or click ⬛ to select it in the drawing.

## Minimize Flat Areas By

Filling Gaps In Contour Data

Specifies that small gaps in contours should be filled in.

Swapping Edges

Specifies that a non-contour common edge shared between a flat triangle and a non-flat triangle should be swapped.

Adding Points To Flat Triangle Edges

Specifies that a new point should be added at the midpoint of an edge shared between a flat triangle and a non-flat triangle.

Adding Points To Flat Edges

Specifies that a new point should be added to edges that bridge two same-elevation data contours and are not in flat triangles.

OK

Click to accept the contour values. You are then prompted at the command line to select objects (Polylines) from the drawing to define as contour data. When the objects are selected, the contour data is added to the surface definition.

**Related procedures:**

■ Weeding and Supplementing Factors for Contours (page 630)

■ Adding Contour Data to a Surface (page 632)

■ Minimizing Flat Areas in a Surface (page 656)

# DEM File (Add/Properties) Dialog Box

Use this dialog box to specify or view the DEM (*.dem*) file and DEM file information.

DEM File Name

Specifies the file name and path for an existing DEM file. Enter the name and path or click 🖳 to open the Import Surface dialog box and browse to the location of the DEM file.

**NOTE** If you are viewing the properties of a DEM file, this field is read only.

DEM File Information

Displays the header information for the selected DEM file. For more information, see About DEM File Data (page 635).

**DEM File**

CS Code

Specifies the coordinate system code for the DEM file. Click ⊡ to select a code in the list of coordinate systems.

Description

Displays the coordinate system description associated with CS Code.

Projection

Displays the coordinate system projection name.

Datum

Displays the coordinate system datum name.

**Current Drawing**

Displays the drawing coordinate system zone information. The drawing coordinate system is specified in the Drawing Settings Dialog Box (page 1851).

CS Code

Displays the current drawings coordinate system code.

Description

Displays the current drawings coordinate system description.

Projection

Displays the current drawing coordinate system projection name.

Datum

   Displays the current drawing coordinate system datum name.

---

**NOTE** The points of the DEM file are transformed from the specified coordinate system of the DEM file to the coordinate system of the current drawing plus any Transformation settings specified in the Drawing Settings dialog box. DEM files cannot be transformed for grid surfaces. If you include a DEM file as part of a grid surface definition, its coordinate system must match that of the drawing, or it cannot be added.

---

**Related procedures:**

■ Adding DEM Files to a Surface (page 637)

■ Creating a Grid Surface from a DEM (page 608)


# Smooth Surface Dialog Box

Use this dialog box to set up the parameters for surface smoothing.

### Smoothing Methods

Select Method

   Specifies the type of surface smoothing to perform:

   ■ **Natural Neighbor Interpolation**: Specifies NNI and disables the Kriging Method property group.

   ■ **Kriging**: Specifies the Kriging method.

### Kriging Method

Specifies the parameters for the Kriging surface smoothing method. Available when the Smoothing Methods property is set to Kriging.

Semivariogram Model

   Specifies the semivariogram to use. For more information, see Smoothing a Surface Using Kriging Method (page 665).

Point Selection Method

   Specifies the points to use for the surface smoothing extrapolation:

   ■ **Select Points**: Selects all points inside a rectangle, polygon, or surface, or parcel.

   ■ **Random Points**: Selects random points inside a rectangle, polygon, or surface, or parcel.

   ■ **Select All Points**: Selects all points on a surface.

Select Points

   Specifies the points that are selected. Available only when either the Select Points or Random Points parameters are selected. Click ⊡ and follow the command line prompts to select the points.

### Point Interpolation/Extrapolation Output

Output Locations

   Specifies the output location for the points:

   ■ **Grid Based**: Outputs points on a grid defined within specified polygon areas selected in the drawing.

   ■ **Centroids**: Outputs points at the existing surface triangle centroids within specified polygon areas selected in the drawing.

   ■ **Random Points**: Outputs a specified number of random points within polygon areas selected in the drawing.

For more information, see Understanding Point Interpolation/Extrapolation Output (page 663).

Select Output Region

Specifies the area which to output the points. Click ⊡ and follow the command line prompts to select the area.

Grid X - Spacing

Specifies the X distance between the grid lines. Available only when the Grid Based option for point output is selected. Click ⊡ to digitize the spacing in the drawing area.

Grid Y - Spacing

Specifies the Y distance between the grid lines. Available only when the Grid Based option for point output is selected. Click ⊡ to digitize the spacing in the drawing area.

Grid Orientation

The orientation direction for the grid in the Y and X directions. Available only when the Grid Based option for point output is selected. Click ⊡ to pick two points in the drawing to define the orientation direction.

Number Of Output Points

Displays the number of points that are output.

**Kriging Semivariogram Model**

This area of the dialog box is displayed if the Kriging surface smoothing method is selected.

Parameter A

Specifies the semivariogram scale factor.

Parameter C

Specifies the semivariogram range factor.

Nugget Effect

Specifies the discontinuity at the origin of the semivariogram model. The magnitude of the discontinuity is called the nugget and must be greater than 0.

Graph display area

Displays the semivariogram model (in red) against the actual surface points (in white). You can visually verify both the model and the surface points at the same time.

**Related procedures:**

■ Smoothing Surfaces (page 662)

# Breakline Properties Vista

Use this panorama vista to view the properties of the breaklines in a surface.

For general information about using panorama vistas, see The Panorama Window (page 102).

The properties grid displays the following columns:

Description

Displays a tree-view of breakline descriptions. Click a breakline to expand it and display its vertices.

Easting (X)

Displays the X coordinate value of a breakline vertex.

Northing (Y)
Displays the Y coordinate value of a breakline vertex.

Elevation (Z)
Displays the Z coordinate value of a breakline vertex.

**Related procedures:**

■ Viewing Breakline Information (page 627)

■ Breaklines (page 618)

# Watershed Display Dialog Box

Use this dialog box to display or hide watersheds by type.

Watershed Type
Specifies whether the watershed type is displayed or hidden. Click the light bulb icon to hide or display the watershed type:

■ 💡 : Specifies that the watershed type is displayed in the analysis and the drawing.

■ 💡 : Specifies that the watershed type is hidden in the analysis and the drawing.

**Related procedures:**

■ Watersheds (page 675)

■ Surface Styles and Visualization (page 692)

# Add Boundaries Dialog Box

Use this dialog box to add boundaries to the surface definition of the selected surface.

Name
Specifies the name of the boundary to be created. If you do not specify a name, the following default naming convention is used: "Boundary<#>." For example, if you do not enter a name, the first boundary is named "Boundary1," the second boundary is named "Boundary2," and so on.

Type
Specifies the type of boundary to add. See Boundaries (page 612):

■ **Outer**: Creates an outer boundary for the surface.

---

**NOTE** Although you can define more than one outer boundary for a surface, only the last one created is used by the surface.

---

■ **Show**: Creates a show boundary. Show boundaries are used to display an area inside a hide boundary.

■ **Hide**: Creates a hide boundary.

■ **Data Clip**: Creates a surface boundary limited by a polygon object from the drawing, such as 2D and 3D polylines, feature lines, survey figures, parcels, and circles. Any data added to a surface following a data clip boundary is clipped to that boundary.

Non-Destructive Breakline

Specifies whether or not to create boundaries with non-destructive breaklines. This clips the triangle edges exactly where they cross the boundary. This option is unavailable when Data Clip is selected as a boundary type.

Mid-Ordinate Distance

Specifies the mid-ordinate distance for the boundaries breaklines, which is used to tessellate the polyline arcs from which the boundary is being created. Click  to digitize the mid-ordinate distance in the drawing area.

**Related procedures:**

■ Boundaries (page 612)

# Boundary Properties Dialog Box

Use this dialog box to view the properties of the boundary in a surface.

Name

Specifies the boundary name.

Type

Displays the boundary type.

**Properties Table**

Displays the following properties:

Vertex

The vertex number of the boundary.

Easting (X)

The X-coordinate value of a boundary vertex.

Northing (Y)

The Y-coordinate value of a boundary vertex.

Elevation (Z)

The Z-coordinate value of a boundary vertex.

**Related procedures:**

■ Editing Boundary Properties (page 617)

■ Adding Boundaries to a Surface (page 615)

# Add Breaklines Dialog Box

Use this dialog box to add breaklines to a surface.

Description

Specifies the description of the breaklines to be created.

Type

Specifies the type of breakline to add:

■ **Standard**: Creates standard breaklines.

- **Proximity**: Creates proximity breaklines.

- **Wall**: Creates wall breaklines.

- **From File**: Imports breaklines from an FLT(.flt) format file.

- **Non-Destructive**: Creates non-destructive breaklines.

For more information, see Types of Breaklines (page 619).

File Link Options

Specifies the file link option to use when importing breaklines from a file:

- **Break Link To File**: Copies breaklines to the surface definition. The file is no longer referenced.

- **Maintain Link To File**: Maintains a reference to the breakline file. The file is used when the surface is rebuilt.

For more information, see Importing Breaklines from a File (page 625).

### Weeding Factors

Available if you select Standard or Wall as the breakline type.

Distance

Specifies the weeding distance (page 630). Enter a value or click  to digitize the mid-ordinate distance in the drawing area.

Angle

Specifies the weeding angle (page 630). Enter a value or click  to digitize the mid-ordinate angle in the drawing area.

### Supplementing Factors

Distance

Specifies the weeding distance. Enter a value or click  to specify the distance in the drawing.

Mid-Ordinate Distance

Specifies the mid-ordinate distance (page 619) for the breaklines. Enter a value or click  to digitize the mid-ordinate distance in the drawing.

**Related procedures:**

- Breaklines (page 618)

# Add Point File Dialog Box

Use this dialog box to specify options before you add a point file to a surface.

Format

Specifies the format of the point data in the file.



Opens the Point File Format (page 2120) dialog box. Select a point file format, edit a point file format, or create a point file format to use when importing the point data.

Source File

Specifies the name of the point data file. Enter a file name, including the full path name.

Opens the Select Source File dialog box. Browse to the location of the point data file. Select the file name, and click Open.

**Advanced Options**

Do Elevation Adjustment If Possible

Specifies that elevation adjustments are performed. The point file format must contain Z+, Z-, or Thickness columns. For more information, see Adjusting Elevation During Import and Transfer (page 499).

Do Coordinate Transformation If Possible

Specifies that coordinate transformations should occur. The point file format must have a coordinate zone assigned to it, and the current drawing must have coordinate zone and transformation information defined. The points are transformed to match the zone of the current drawing.

Do Coordinate Data Expansion If Possible

Specifies that coordinate data properties of the points, such as degrees, minutes, seconds, and hemisphere for latitude and longitude, should be calculated if possible. These values are calculated from known coordinate data information contained in the point data file, such as grid northing and grid easting.

**Related procedures:**

■ Adding Surface Data from a Point File (page 643)

# Add Points from Drawing Objects Dialog Box

Use this dialog box to add point data from AutoCAD objects to the surface definition.

Object Type

Specifies the type of AutoCAD object to add:

■ **Points**: Creates surface point data from AutoCAD Point objects. The points XYZ coordinates are used to define the surface point.

■ **Lines**: Creates point data from AutoCAD Line objects. The XYZ coordinates of the objects endpoints are used to define surface points.

■ **Blocks**: Creates point data from AutoCAD Block Reference objects. The block insertion point XYZ coordinates are used to define the surface point.

■ **Text**: Creates point data from AutoCAD Text objects. The text insertion point XYZ coordinates are used to define the surface point.

■ **3D Faces**: Creates point data from AutoCAD 3DFace objects. The XYZ coordinates of the objects endpoints are used to define surface points.

■ **Polyface**: Creates point data from AutoCAD PolyFaceMesh objects. The XYZ coordinates of the objects endpoints are used to define surface point.

Maintain Edges From Objects

Specifies whether to define the AutoCAD Civil triangle edges based on the edges defined in the original AutoCAD object. If selected, AutoCAD Civil maintains the edges and does not attempt to optimize the edges. Available when importing points from lines, 3D faces, and polyfaces.

Description

Specifies the description for the point data to be created.

■ [Adding Surface Point Data from AutoCAD Drawing Objects](#) (page 640)

# Create Mask/Mask Properties Dialog Box

Use this dialog box to create a mask or to modify the properties of an existing mask.

### Information

Name
Specifies the name of the mask.

Description
Specifies the description of the mask.

### Masking

Mask Type
Specifies the mask type:

■ **Inside**: Creates an inside mask, which clips and hides the area inside the parcel or polygon area.

■ **Outside**: Creates an outside mask, which clips and hides the area outside the parcel or polygon area.

Render Only
Specifies whether you want to create a render-only mask with the specified render material applied to the inside of the parcel or polygon.

Mid-Ordinate Distance
Specifies the mid-ordinate distance for the region segments, which is used to tessellate the polyline arcs from which the region is being created. Click  to digitize the mid-ordinate distance in the drawing area.

Render Material
Specifies the render material (page 1577). Click  to select a render material.

### Related procedures:

■ [Masks](#) (page 669)

# Mask Display Order Dialog Box

Use this dialog box to control the display order for render-only masks.

Masks listed higher are displayed on top of masks that are listed lower.

### Mask List

This area lists the existing masks. Click a mask to select it.

Name
Displays the name of the mask.

Description
Displays the description for the mask.

⇧

Click to move the selected mask up in the display order list.

Click to move the selected mask down in the display order list.

**Related procedures:**

■ Changing the Display Order of Render Only Masks (page 673)

# Composite Volume Vista (Panorama)

Use this panorama vista to create composite volume calculations.

 Create New Volume Entry

Creates a new composite volume entry.

 Delete Volume Entry

Deletes the highlighted composite volume entry from the list.

 Import Volume Entries

Imports a volume entry that has been saved in XML format. Click to open the Import Volume Entries dialog box, where you can browse for the XML file. After you select the file, a new volume entry is created in the list.

 Export Volume Entries

Exports the selected volume entry to an XML file. Click to display the Export Volume Entries dialog box, from which you enter the file name and can browse to a location to save the XML file.

 Recompute Volumes

Recalculates the composite volume. For example, if you have created a composite volume on a surface that has been updated, you can click this button to recalculate the volume.

 Create New Volume Entry from Surfaces

Creates a new composite volume entry. You are prompted to select the base surface (Surface 1) and comparison surface (Surface 2). The cut, fill, net, and net graph are calculated and displayed in the table.

Index

Displays a list of numerical identifier assigned to each composite volume calculation.

**Surface Pair**

Base Surface

Specifies the base surface from which to create the composite volume. Click on the field and select the surface from the list.

Comparison Surface

Specifies the comparison surface from which to create the composite volume. Click on the field and select the surface from the list.

**Volume**

Cut

Displays the amount of material that has to be removed for the base surface to equal the comparison surface.

Fill

Displays the amount of material that has to be added for the base surface to equal the comparison surface.

Net

Displays the cut minus the fill. For example, if a volume is 200 m3 of cut, and 100 m3 of fill, the net is 100 m3<cut>.

Net Graph

Displays a graphical percentage representation of the whole volume. A fill net is displayed as a green bar indicating that material needs to be added to the project site. A cut net is displayed as a red bar, indicating that material must be removed.

**Related procedures:**

■ [Calculating Composite Volumes](#) (page 690)

# Hatch Properties Dialog Box

Use this dialog box to select the display of the watershed hatching pattern.

Type

Sets the pattern type. Click the ![button] button to select the hatching type from the drawing area:

■ **Predefined**: Specifies a predefined AutoCAD pattern. These patterns are stored in the *acad.pat* and *acadiso.pat* files. You can control the angle and scale of any predefined pattern. For predefined ISO patterns, you can also control the ISO pen width.

---

**NOTE** When you use the Solid predefined pattern, ensure that the boundary is closed and does not intersect itself. In addition, if the hatch area contains more than one loop, the loops must not intersect. These limitations do not apply to standard hatch patterns.

---

■ **User-Defined**: Creates a pattern of lines based on the current linetype in your drawing. You can control the angle and spacing of the lines in your user-defined pattern.

■ **Custom**: Specifies a pattern that is defined in any custom PAT file that you have added to the AutoCAD search path. (To use the patterns in the supplied *acad.pat* and *acadiso.pat* files, choose Predefined.) You can control the angle and scale of any custom pattern.

Pattern

Lists the available predefined patterns. The six most recently used predefined patterns appear at the top of the list. The selected patterns are stored in the HPNAME system variable. Available only if Type is set to Predefined.

Click ![button] to open the Hatch Pattern Palette dialog box, where you can view preview images for all predefined patterns.

Custom Pattern

Lists the available custom patterns. The six most recently used custom patterns appear at the top of the list. AutoCAD stores the selected pattern in the HPNAME system variable. Available only if Type is set to Custom.

Click ![button] to open the Hatch Pattern Palette dialog box, where you can view preview images for all custom patterns.

Angle

Specifies an angle for the hatch pattern relative to the X axis of the current UCS. AutoCAD stores the angle in the HPANG system variable.

ISO Pen Width

Specifies that an ISO predefined pattern is scaled based on the selected pen width. Available only if you set Type to Predefined and set Pattern to one of the available ISO patterns.

Scale

Expands or contracts a predefined or custom pattern. AutoCAD stores the scale in the HPSCALE system variable. This option is available only if you set Type to Predefined or Custom.

Relative to Paper Space

Specifies that the hatch pattern is scaled relative to paper space units. Using this option, you can easily display hatch patterns at a scale that is appropriate for your layout. Available only from a layout.

Spacing

Specifies the spacing of lines in a user-defined pattern. AutoCAD stores the spacing in the HPSPACE system variable. Available only if you set Type to User-Defined.

**Related procedures:**

■ Watersheds (page 675)

# Select Coordinate Zone Dialog Box

Use this dialog box to select the coordinate system.

**Zone**

Categories

Specifies the category of the coordinate system. This constrains the available coordinate systems.

Available Coordinate Systems

Specifies the available coordinate systems. Select from the list or enter a known coordinate system.

Selected Coordinate System Code

Specifies the code for the coordinate system. Populated when you specify a coordinate system or you can enter a coordinate system code.

Description

Displays a description for the selected coordinate system.

Projection

Displays the projection for the selected coordinate system. For example, LL for Latitude/Longitude or TM for Transverse Mercator.

Datum

Displays the datum for the selected coordinate system.

**Related procedures:**

■ Creating a Grid Surface from a DEM (page 608)

# Select Render Material Dialog Box

Use this dialog box to specify a render material for an object.

Render materials list

Select a render material from the list.

Click to select a render material from the drawing.

**Related procedures:**

■ [Applying Render Materials to Objects](page 1578)

# Water Drop Dialog Box

Use this dialog box to specify the parameters for drawing water drop paths on a surface.

---

**NOTE** The default values for these parameters can be set in the Edit Command Settings - CreateSurfaceWaterdrop dialog box. For information about setting command parameters, see [Specifying Command-Level Settings](page 69).

---

## Water Drop Path

Path Layer

Specifies the layer on which to draw the water drop path. Click  to open the [Object Layer](page 1985)dialog box where you can select a layer.

Path Object Type

Specifies the type of AutoCAD object to use for the water drop path. Either 2D Polyline or 3D Polyline.

## Water Drop Marker

Place Marker At Start Point

Specifies whether to draw a marker at the start point of the water drop path.

Start Point Marker Style

Specifies the style for the start point marker. Click  to open the [Select Point Style](page 1801) dialog box where you can select a style.

**Related procedures:**

■ [Drawing Water Drop Paths](page 711)

# Minimize Flat Areas Dialog Box

Use this dialog box to specify the parameters for minimizing flat areas on a surface.

## Minimize Flat Areas By

Filling Gaps In Contour Data

Specifies that small gaps in contours should be filled in.

Swapping Edges

Specifies that a non-contour common edge shared between a flat triangle and a non-flat triangle should be swapped.

Adding Points To Flat Triangle Edges

Specifies that a new point should be added at the midpoint of an edge shared between a flat triangle and a non-flat triangle.

Adding Points To Flat Edges

Specifies that a new point should be added to edges that bridge two same-elevation data contours and are not in flat triangles.

**Related procedures:**

■ [Minimizing Flat Areas in a Surface](#) (page 656)

# Export Surface to DEM Dialog Box

Use this dialog box to specify the parameters for exporting a surface to a DEM file.

## Selected Surface

Name

Displays the name of the selected surface.

Description

Displays the description for the selected surface.

Drawing Coordinate Zone

Displays the drawing coordinate zone, if specified.

## Export

DEM File Name

Specifies the file name, location, and file type of your new DEM file. The file can be of type USGS (*.dem*) or GEOTIFF (*.tiff*). Click  to open the Export Surface to DEM dialog box, then browse to the location, enter the file name and select file type.

Export Coordinate Zone

Specifies the coordinate zone of the exported surface. If the drawing has a coordinate zone assigned, this coordinate zone is displayed. Click  to open the Select Coordinate System dialog box to select the required coordinate system from the list.

Grid Spacing

Specifies the horizontal spacing for the DEM profile points. Enter an integer value that is greater than or equal to 1. This value is specified in units used in the export coordinate zone.

Determine Elevations By

Specifies how the elevations of the DEM file are determined from the exported surface. Select one of the following values:

■ Sample surface at grid point: Ensures that for each point along a DEM profile, the elevation is sampled from the selected surface.

■ Average: Ensures that for each point along a DEM profile, the elevation value corresponds to the mean elevation in a grid cell-sized region centered at this point.

Use Custom Null Elevation

Specifies whether to use a custom value for null elevation.

Null Elevation

Specifies the default value for null elevation.

**Related procedures:**

■ [Exporting to DEM](#) (page 714)

# Select Coordinate System Dialog Box

Use this dialog box to specify a coordinate system for an exported DEM file.

Drawing Coordinate System

Displays the coordinate system used in the drawing that references the surface.

Select DEM System

Specifies the available coordinate systems in the Projected category. Click to see the complete list of Projected coordinate systems.

Code

Displays the code for the selected coordinate system.

Area Of Use

Displays the region of the world where the selected coordinate system is used.

Properties

Click to see the detailed properties of the selected coordinate system and its projection.

**Related procedures:**

■ Exporting to DEM (page 714)

# DEM Coordinate System Properties Dialog Box

Use this dialog box to view the detailed properties of the coordinate system specified in the Select Coordinate System dialog box.

# Extract Objects from Surface Dialog Box

Use this dialog box to extract AutoCAD objects from a surface.

Property

Displays the currently visible surface components based on the surface style settings (see Display Tab (Surface Style Dialog Box) (page 2345). By default, all the displayed surface components are selected when the dialog box is opened.

Value

Specifies one of the following options:

■ Select All Extracts all the surface objects of the selected type.

■ Select from Drawing  Extracts an individual surface element. Click  to select the object you want to extract from the drawing.

**Related procedures:**

■ Extracting Surface Data (page 646)

# Move Blocks to Surface Dialog Box

Use this dialog box to move selected Block Reference objects to the elevation on the selected surface.

**Select block reference names**

Displays the names of each block definition that is referenced in the drawing. Select a block definition name from the list to select all Block Reference objects in the drawing or click  to select individual Block Reference objects and then press Enter.

**Total block reference objects selected**

Displays the number of Block Reference objects selected.

**Related procedures:**

■ Moving Blocks to a Surface Elevation (page 645)

# Move Blocks to Attribute Elevation Dialog Box

Use this dialog box to move selected Block Reference objects to a selected attribute elevation.

**Select block reference name**

Use to select a block definition name from the list. Click  to select individual Block Reference objects from the drawing and then press Enter.

**Select elevation attribute tag**

Specifies attribute tags for the selected Block Reference objects. If the Block Reference object does not have associated attribute tags, this option is unavailable.

**Total block reference objects selected**

Specifies the number of Block Reference objects selected.

**Related procedures:**

■ Moving Blocks to Attribute Elevation (page 641)

# Drape Image Dialog Box

Use this dialog box to select an image from the drawing and overlay it over a surface.

**Image**

Specifies the image from the drawing to be overlaid on the surface. Click  to select the image.

**Surface**

Specifies the surface on which the image is draped.

**Render Material Name**

Specifies the name of the render material associated with the current image. Once created, this render material can be applied to a different suitable surface.

**Related procedures:**

■ Draping Images On Surfaces (page 695)

# Simplify Surface Wizard

Use this wizard to reduce the number of points in a TIN surface.

# Simplify Methods Page (Simplify Surface Wizard)

Specify the surface simplification methods.

Edge Contraction

Contracts triangle edges to single points. The location of the point to which an edge is contracted is chosen so that the change to the surface is minimal. Each edge contraction results in the removal of one point.

Point Removal

Selects and removes surface points. More points are removed from the denser areas of the surface.

**Related procedures:**

■ Simplifying Surfaces (page 666)

# Region Options Page (Simplify Surface Wizard)

Specify surface regions for simplification.

**Region Options**

Use Existing Surface Border

Specifies the existing surface border as the boundary of the surface simplification region.

Specify Window/Polygon

Specifies the surface simplification region within a drawn rectangle.

Pick In Drawing

Click to specify the first and second points of the rectangle that delimits the surface simplification region. The button is unavailable, if Use Existing Surface Border is selected as a simplification option.

Select Objects

Specifies one of the following surface objects: 2D or 3D polyline, parcel, circle, feature line, or survey figure.

Mid-Ordinate Distance

Specifies the default mid-ordinate length used for the Select Objects option. If the selected object contains arc segments, each segment is tessellated into chord segments. The length of each chord segment is derived from the specified mid-ordinate distance. See also Mid-ordinate distance (page 615). Enter the distance value or click  to select the distance in the drawing.

Pick In Drawing

Click to select an object for simplification. The button is unavailable if Use Existing Surface Border is selected as a simplification option.

Total Points Selected In Region

Specifies the total number of points in the selected surface simplification region.

**Related procedures:**

■ Simplifying Surfaces (page 666)

# Reduction Options Page (Simplify Surface Wizard)

Fine-tune surface point reduction options.

### Reduction Options

Total Points Selected In Region

Specifies the total number of points in the selected surface simplification region.

Percentage Of Points To Remove

Specifies the percent of surface points to remove.

Maximum Change In Elevation

Specifies the maximum allowed difference between the elevation of the original surface and the elevation of the simplified surface.

Maximum Edge Contraction Error

Appears if you select Edge Contraction as a surface simplification method. It specifies a degree of the surface change if the edge were contracted to a point. 100 represents the maximum error over all surface edges and 0 represents the minimum.

Total Points Removed

Specifies the number of points removed after surface simplification. Press Apply to display the number.

**Related procedures:**

■ Simplifying Surfaces (page 666)

# Catchment Area Dialog Box

Specify the properties of a catchment boundary and catchment point marker.

### Catchment Area

Catchment Layer

Specifies the layer for the catchment region. The default value is zero. Click  in the Value column to select a different layer.

Catchment Object Type

Specifies the object type for the catchment boundary.

■ Select 2D Polyline if you simply want to delineate a catchment region.

■ Select 3D Polyline if you want to perform visualization and drape the catchment boundary line over the surface.

Catchment Marker

Specifies whether or not to add a catchment point marker to a catchment region.

Catchment Marker Style

Specifies the style of the catchment point marker. The default style is Standard. Click  to specify a different style.

**Related procedures:**

■ Displaying and Calculating Catchment Areas (page 712)

# Survey Dialog Boxes

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## Edit Feature Settings - Survey Dialog Box

Use this dialog box to view and change survey-specific settings.

This topic documents settings in all survey-related Edit Settings dialog boxes (drawing-level, feature-level, and command-level).

- Drawing-level ambient settings are identified by the ⬚ drawing icon.

- Survey feature settings are listed near the top of this dialog box, after the General property group, and are identified by the 📷 survey icon.

- Survey command settings are identified by the 🗹 command icon.

For general information about drawing, feature, and command settings and their interaction, see Working with the Standard Settings Dialog Box Controls (page 62).

For information about drawing-level ambient settings, see Ambient Settings Tab (Drawing Settings Dialog Box) (page 1856)

### 📷 Default Styles

Specify the default styles for the survey features.

Default Figure Style

   Specifies the default style for a survey figure. Click in the Value column, and click ⬚ to select a style in the Survey Default Figure Style dialog box

Default Network Style

   Specifies the default style for a survey network. Click in the Value column, and click ⬚ to select a style in the Survey Default Network Style dialog box

For information about style selection, see Select Style Dialog Box (page 1805).

# Survey User Settings Dialog Box

Use this dialog box to change and view survey user settings.

Survey user settings are specific to a Windows user login account and affect the survey features, not database, or drawing data.

**Miscellaneous**

Establish default external editor settings.

Use External Editor

Specifies whether to use an external editor for displaying analysis input and output and editing field book and batch file. Click the check box to use an external editor.

> **NOTE** If you do not specify to use an external editor, the default editor, specified in the AutoCAD Options dialog box, is used.

External Editor

Specifies the path and name of the external editor. Enter a name and path or click ⌕ to browse to a location.

Preview Vertical Exaggeration

Specifies if the network, setup, and figures previews are exaggerated vertically. Enter a value, which is the factor applied to the elevations of the objects displayed in the previews.

**Survey Database Defaults**

Establish default locations for the survey database settings.

Survey Database Settings Path

Specifies the path for the location of the survey database settings files. The .*sdb_set* files located in this path can be selected in the Survey Database Settings.

Survey Database Settings

Specifies the survey .*sdb_set* file. This file contains the default database settings that are used when you create a new survey database.

Extended Properties Definition Path

Specifies the path for the location of the survey extended properties definition files. The .*sdx_set* files located in this path can be selected in the Extended Properties Definition.

Extended Properties Definition

Specifies the .*sdx_set* file. This pre-defined schema contains both LandXML and user defined properties that can be specified as the default when you create a new survey database.

**Equipment Defaults**

Establish default equipment database settings.

Equipment Database Path

Specifies the path for the equipment databases. This is the path where all new equipment databases are stored. Enter a path or click ⌕ to browse to a location.

Current Equipment Database

Specifies the current equipment database. Click the equipment database from the drop-down list.

> **NOTE** The list of available databases is determined by the databases listed in the Equipment Databases collection in the Toolspace Survey tab.

Current Equipment

Specifies the current equipment to use. Click the equipment name from the drop-down list.
The current equipment sets the values associated with a specific surveying instrument, such as the standard
deviations associated with the measuring capabilities for the equipment. This information is used in
various calculations including least squares.

## Linework Processing Defaults

Establish the default location for the linework code set and specify the linework code set to be used to do
linework processing during the import.

Linework Code Sets Path

Specifies the path for the Linework Code Sets. This is the path where all linework code sets are stored.

Enter a path, or click ⬛ to browse to a location.

Process Linework During Import

Specify Yes to process linework during an import command.

Current Linework Code

Specifies the default linework code set to use when linework processing occurs.

Process Linework Sequence

Specifies the order in which survey point descriptions are processed to determine linework connectivity.
Select one of the following:

■ By Import Order - processes points in the order in which they are imported. Point names are always
processed by import order.

■ By Point Number - processes points sequentially by point number (ascending order only).

## Figure Defaults

Establish equipment and figure prefix database information as well as the default external editor.

Figure Prefix Database Path

Specifies the path for the figure prefix database. This is the path where all new equipment databases are

stored. Enter a path or click ⬛ to browse to a location.

Current Figure Prefix Database

Specifies the current figure prefix database. Click the figure prefix database from the drop-down list.
Figure prefixes enable you to determine the layer that a figure is drawn on by creating a prefix for a group
of figure names. All figures with a specific prefix are placed on a specific layer.

NOTE The list of available databases is determined from the databases listed in the Figure Prefix Databases
collection on the Toolspace Survey tab.

Figure Style

Specifies the default style for figures inserted into the drawing.

Figure Layer

Specifies the default layer for figures inserted into the drawing.

## Interactive Graphics

Control the display of survey components during import and entry of survey data.

NOTE To change the colors for the components, click the color swatch to open the Select Color dialog box, from
which you can select a color.

Automatic Pan and Zoom

Specifies if the drawing automatically pans and zooms to the survey observations when they are imported or entered.

Show Backsight Line

Specifies whether a line, which represents the line backsighted by the survey instrument setup, is displayed.

Show Backsight Prism

Specifies whether a symbol, which represents the prism at the backsight point, is displayed.

Show Station Instrument

Specifies whether a symbol, which represents the instrument (transit) at the station (or occupied) point, is displayed.

Show Foresight Line

Specifies whether a line, which represents the line foresighted by the survey instrument setup, is displayed.

Show Foresight Prism

Specifies whether a symbol, which represents the prism at the foresight point, is displayed.

Show Baseline

Specifies whether a line, which represents the line established as a baseline, is displayed.

Show Baseline Prism

Specifies whether a symbol, which represents the prism at any new point created from a baseline, is displayed.

Show Baseline Offset Line

Specifies whether a line, which represents the baseline offset to any new point created, is displayed.

Show Current Figure

Specifies whether a line, which represents the current figure, is displayed.

## Import Defaults

Specify the properties for importing a batch file into the selected survey network.

Show Interactive Graphics

Specifies whether to display interactive graphics during the import of survey data. Use Interactive Graphics settings to control the display of individual graphics components.

Erase Survey Points From Drawing

Specifies whether to erase all the points referenced by the selected network from the drawing.

**NOTE** Points referenced by other networks are not erased from the drawing and database.

Reset Network

Specifies whether to remove all the observations in the survey database for the selected network. When this property is unchecked, incoming observations are appended to the observation database for the selected network.

Delete Network Figures

Specifies whether to erase figures, which have a reference to the selected network, from the drawing and database.

Insert Network Object

Specifies whether to create a network object in the drawing.

Insert Figure Objects

Specifies whether to create figure objects in the drawing.

Insert Survey Points

Specifies that Survey points are inserted into the drawing.

Default Figure Site

Specifies the default site for figures that are set to create Lot Lines. When inserting a figure into the current drawing:

■ If this site does not exist in the current drawing, it is created.

■ This site is used if the Lot Line property for the figure is set to True, and the Site property for the figure is blank.

Display Tolerance Errors In Event Viewer

Select Yes to display tolerance errors in the Event Viewer when you import data using the Import Field Book, Import LandXML, or Import Batch File commands. For more information, see Reporting Tolerance Errors (page 211).

## Export Defaults

Specify the properties for exporting survey data to a field book file.

Export Figures With Network

Select Yes to export the figures in the database when you use the Export Field Book command.
The figure type determines what is added to the field book. For information on the conditions and syntax that is added to the field book, see Setting Export Defaults (page 213).

Export Point Identifiers

Specifies if point IDs are used to define figures when figures are exported to a field book.
For information on the conditions and syntax that is added to the field book, see Setting Export Defaults (page 213).

Export Point Data

This check box is active when the Export Point Identifiers property is not set to None. Select Yes if you want each figure vertex that references a point identifier to output the following to the field book file prior to the section defining the figures: NE SS <point ID> <north> <east> <elevation> <description>

## Network Preview

Specify the network preview properties when selecting a Network item in the Survey Toolspace tab.

Show Network

Specifies whether to preview the network lines for the selected network item.

Show Sideshots

Specifies whether to preview the sideshot lines for the selected network item.

Show Points

Specifies whether to preview the points for the selected network item.

Show Figures

Specifies whether to preview the figures that reference the selected network item.

## Setup Preview

Specify the setup preview properties when selecting a Setup item in the Survey Toolspace tab.

Show Sideshots

Specifies whether to preview the sideshot lines for the selected setup item.

Show Points

Specifies whether to preview the points for the selected setup item.

Show Figures

Specifies whether to preview the figures affected by the selected setup item.

**Figure Preview**

Specify the setup preview properties when selecting a Figure item in the Survey Toolspace tab.

Show Figure

Specifies whether to preview the line segments for the selected figure item.

Show Points

Specifies whether to preview the display of the points referenced by the selected figure item.

**Related procedures:**

■ User Settings (page 205)

# Survey Database Settings Dialog Box

Use this dialog box to modify the survey database settings. These settings are specifically for the survey database features. By default, the survey database settings are stored in C:\Civil Projects\<database name>\Survey.sdb file.

**Units**

Establish specify the database units.

Coordinate Zone

Click ⬚ to display the Select Coordinate Zone dialog box where you can specify the coordinate zone for the survey database data. The default zone is set to None.

Distance

Specifies the distance unit. Click the cell in Value column and select a distance unit from the list.

**NOTE** If you specify a Coordinate Zone, the distance unit is obtained from the zone and the Distance property field is not active.

Angle

Specifies the angle unit for the survey database data. Click the cell in the Value column and select an angle type from the list.

Direction

Specifies the direction type for the Survey database data. Click the cell in the Value column and select a direction type from the list.

Temperature

Specifies the temperature unit for the survey database data when temperature corrections are applied to observations. Click the cell in the Value column and select a temperature type from the list.

Pressure

Specifies the atmospheric pressure unit for the survey database when pressure corrections are applied to observations. Click the cell in the Value column and select a pressure type from the list.

**Precision**

Establish the display precision of the survey data.

**NOTE** These precision settings are independent of the Drawing Settings precision settings. These settings affect all aspects of the user interface that displays survey data.

Angle

Specifies the precision value for all angle (including direction) values. The default precision is 4.

Distance

Specifies the precision value for all distance values. The default precision for Meters is 3, the default precision for Feet is 2.

Elevation

Specifies the precision value for all elevation values. The default precision for Meters is 3 and Feet is 2.

Coordinate

Specifies the precision value for all coordinate values. The default precision is 4.

Latitude and Longitude

Specifies the precision value for all latitude and longitude values. The default precision is 8.

## Measurement Type Defaults

Establish the default types assigned to the measurements.

**NOTE** For more information about the measurement types, see the Setting Measurement Type Defaults (page 217).

Angle Type

Specifies the default angle type. Click the cell in the Value column and select a distance type from the list.

Distance Type

Specifies the default distance type. Click the cell in the Value column and select a distance type from the list.

Vertical Type

Specifies the default vertical type. Click the cell in the Value column and select a vertical type from the list.

Target Type

Specifies the default target type. Click the cell in the Value column and select a target type from the list.

## Measurement Corrections

Specify the default measurement corrections to apply to the survey observations.

**NOTE** For information on formulas used to calculate corrections, see Setting Measurement Corrections (page 218).

Curvature and Refraction

Select Yes to correct measurement calculations for curvature of the earth and refraction of the sun light.

Sea Level

Select Yes to correct measurement calculations to sea level.

Atmospheric Conditions

Select Yes to correct measurement calculations for temperature and pressure.

Horizontal Collimation

Select Yes to correct Face 1 Angle and Face 2 Angle types for horizontal collimation (page 2470).

Vertical Collimation

Select Yes to correct Face 1 Angle and Face 2 Angle types for vertical collimation.

Scale Factor

Select Yes to correct calculated horizontal distances for scale factor.

EDM-Prism Eccentricity

Select Yes to correct distance measurements for EDM (page 2473) and prism (page 2480) offsets.

## Traverse Analysis Defaults

Do Traverse Analysis

Select Yes to perform a mathematical analysis based on the method you specify as the Horizontal Adjustment Method.

Do Angle Balance

Select Yes to have the angular error evenly distributed throughout the traverse.

Horizontal Adjustment Method

Specify one of the following traverse analysis methods:

- Compass Rule: A method of corrections where the closing errors are assumed to be as much due to errors in observed angles as by the errors in measured distances. The closing errors in latitude and departure are distributed according to the ratio of the length of the line to the total length of the traverse.

- Transit Rule: A method of balancing a traverse where the closing errors are assumed to be caused less by the errors in the observed angles than by the errors in measured distance. Corrections are distributed according to the ratio of the latitude and departure of each leg of the traverse to the sums of the latitude and departures of the entire traverse.

- Crandall Rule: A method of balancing a traverse where all the angular error is distributed throughout the traverse and all adjustments to the traverse are due to modifying the traverse distances. The modification distance made to each leg is such that the sum of the squares is a minimum. Corrections corresponding to the closing errors assume that the closing errors are random and normally distributed, and that all the angular error has been adjusted prior to the adjustment routine.

- Least Squares: A method of balancing a traverse. The squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method weights the individual measurements according to the specifications set in the Equipment correction settings to determine the source of error. You can adjust the data for an individual traverse loop, or a traverse network, if located from multiple observations.

Vertical Adjustment Method

Specify one of the following vertical adjustment methods:

- None: No vertical adjustment performed.

- Length Weighted Distribution: Vertical adjustment distributes the vertical closing error to each line at the same ratio as the length of the line to the total length of the traverse (similar to the Compass rule).

- Equal Distribution: Vertical adjustment distributes the vertical closing error equally to each of the traverse stations.

- Least Squares: A method of distributing the vertical error throughout a traverse. The squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method weights the individual measurements according to the specifications set in the Equipment correction settings to determine the source of error. You can adjust the data for an individual traverse loop, or a traverse network, if located from multiple observations.

**NOTE** The Least squares vertical adjustment method is only available if Least squares has been selected as the horizontal adjustment method. When this method is selected, a 3D least squares adjustment is performed. If this method is not selected a 2D least squares adjustment is performed.

Horizontal Closure Limit 1:X

Specifies the minimum allowable error of closure in the horizontal direction, for example a value of 15000.00 is 1 part in 15000.

Vertical Closure Limit 1:X

Specifies the minimum allowable error of closure in the vertical direction, for example a value of 15000.00 is 1 part in 15000.

Angle Error Per Set

Specifics the maximum angular error per traverse observation set.

## Least Squares Analysis Defaults

Specify the survey database setting defaults for performing a least square analysis on a network or a traverse.

Network Adjustment Type

Specifies the adjustment type for network least squares analysis only. The Traverse Analysis Defaults settings determine either a 2D or 3D analysis. Click the cell in the Value column and select one of the following from the list:

■ 2-Dimensional: Sets the adjustment type to 2D. No adjustments are made to elevations. Adjustments are only made to horizontal positions.

■ 3-Dimensional: Sets the adjustment type to 3D. Adjustments are made to both horizontal and vertical positions.

Maximum Number of Iterations

Specifies the maximum number of times the least squares routine adjusts the observations before a solution is reached. This default setting is 8. The maximum number of iterations that you can enter is 10. Generally, the solution is reached within two or three iterations.

Coordinate Convergence

Specifies the maximum difference that is allowed between the coordinates before the solution is accepted.

Confidence Level

Specifies either 95% or 99% as the percentages calculated in conjunction with the F distribution. F is the ratio of two independent chi square variables, which are divided by the respective degrees of freedom. For example, if the confidence level is set to 99%, then you can be 99% sure that the calculated ellipse contains the true location for the point.

Perform Blunder Detection

Select Yes to include blunder detection in the least squares output (*.lso) file.

## Survey Command Window

Specify the survey database settings for interacting with the Survey Command Window.

Ditto Feature

Select Yes to automatically repeat the last command in the Survey Command Window. This setting is on by default. When this setting is on, you can enter the command name once, and then continue to use the command by entering numeric data. To end the command, enter another command name.

Auto Point Numbering

Select Yes to enable automatic point numbering. When automatic point numbering is on, new points are automatically assigned the next available point number. When automatic point numbering is off, you must specify new point numbers as they are needed.

Start Point Numbering From

Specifies the number at which to start Auto Point Numbering.

Point Course Echo

Select Yes to have the Survey Command Window report the course taken to reach a point, including the direction and the distance.

Affects output in both the output file and the output view of the Survey Command Window.

Figure Course Echo

Select Yes to have the Survey Command Window report the course taken to create a figure, including direction and distance.

Affects output in both the output file and the output view of the Survey Command Window.

Point Coordinate Echo

Select Yes to have the Survey Command Window display the point coordinates, including northing, easting, and elevation information.

Affects output in both the output file and the output view of the Survey Command Window.

Figure Coordinate Echo

Select Yes to have the Survey Command Window display figure point coordinates, including northing, easting, and elevation information.

Affects output in both the output file and the output view of the Survey Command Window.

Command Echo

Select Yes to have the Survey Command Window display the commands that are entered.

Affects output in both the output file and the output view of the Survey Command Window.

Use Batch File

Select Yes and specify a batch file to record Survey Command Window input per network. The batch file for each network in a survey database is saved to *Civil Projects\<named> Survey Database\<named> Network\Batch.txt*.

If set to No, no commands are displayed in the Batch view of the Survey Command Window.

Use Output File

Select Yes and specify an output file name to record Survey Command Window output. The output file for each network in a survey database is saved to *Civil Projects\<named> Survey Database\<named> Network\Output.txt*.

## Error Tolerance

**NOTE** Redundant observations that exceed the error tolerance values create an event in the Event Viewer. For more information, see Reporting Tolerance Errors (page 211).

Distance Difference

Specifies a maximum difference in distance. If you observe a distance to a point more than once and the difference is greater than the acceptable value then an event is listed in the Event Viewer and a warning icon is displayed on the observation in Toolspace. If the difference in distance is less than the acceptable value, then the distance difference is averaged to establish the point coordinates.

Angular Difference

Specifies the maximum angular difference. If you observed an angle to a point more than once and the difference is greater than the acceptable value then an event is listed in the Event Viewer and a warning icon is displayed on the observation in Toolspace. If the angular difference is less than the acceptable value, then the angular difference is averaged to establish the point coordinates.

Elevation Difference

Specifies the maximum elevation difference. If you observed an elevation to a point more than once and the difference is greater than the acceptable value then an event is listed in the Event Viewer and a warning icon is displayed on the observation in Toolspace. If the elevation difference is less than the acceptable value, then the elevation difference is averaged to establish the point elevation.

Coordinate Difference

Specifies the maximum coordinate difference. If you located a point from more than one setup, and the difference is greater than the acceptable value then an event is listed in the Event Viewer and a warning icon is displayed on the observation in Toolspace. If the coordinate difference is less than the acceptable value, then it is averaged to establish the point coordinates.

## Extended Properties

Specify the survey database extended property settings.

Create New Definitions Automatically

Specify Yes to automatically create Survey LandXML properties in the LandXML file that do not exist in the Survey database.

Display Warnings For Missing Required Properties

Specify Yes to display a warning in the Event Viewer for any instance of a <SurveyFeatureClass> that is missing a required property. The Event Viewer tree displays the survey database name and the command name. The Event Viewer description column displays the <SurveyFeatureClass><ObjectID>:Missing required <property name> value.

**Related procedures:**

■ Survey Database (page 196)

# Survey Tab Item View

Use the Survey tab item view to display information about the contents of an individual item or a collection and to access related commands.

For some collections and some individual items, you can display additional information in an item view, which is displayed in a pane next to the tree when the Toolspace window is floating and beneath the tree when the Toolspace window is docked. To display the item view for a collection or an individual item in the tree, click the item name.

---

**NOTE** To see the item view, you may need to increase the size of the Toolspace window or move the bar that separates the item view from the tree. To move the bar that separates the item view from the tree, you must first select a collection that displays an item view.

---

An item view can be a collection list view, properties list view, or graphical view, depending on the selected item.

If there are editable fields in the list view, the following icons are displayed:

Discards any edits made to the properties and reloads the properties from the relevant database.

Saves any edits made to the properties back to the relevant database.

Bold text in the survey item view indicates that there are unsaved changes made to data. Click to save the changes or to discard the changes and revert the data.

## LandXML

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file that are relevant to networks are displayed. Click the cell in the Value column to assign an attribute.

### User-defined

If you are using the extended properties feature, and have created User-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

### Collection List View

A collection list view is a grid of rows and labeled columns displayed by the Toolspace trees for collection items. The list view can be displayed beneath or next to the Toolspace tree, depending on the state of the Toolspace window. The data displayed in the list view depends on the item selected in the tree.

### Properties List View

A properties list view displays an editable view of the properties of the item selected, for example, and individual figure, control point, or setup.

### Graphical View

A graphical view displays information about the selected item in a graphical (picture) form.

You can right-click in the graphical preview to display a shortcut menu that allows you to manipulate the image using commands such as Pan and Zoom.

### Related procedures:

- The Toolspace Item View (page 82)
- The Toolspace Survey Tab (page 97)

# New Equipment/Equipment Properties Dialog Box

Use this dialog box to change and view survey equipment properties.

### Miscellaneous

Name

Specifies the name of the equipment that displays in the selected equipment database.

> **NOTE** A best practice is to use the model number or the name of the equipment manufacturer.

Description

Specifies a description of the surveying equipment.

### Units

Distance

Specifies the type of distance units to be used for the equipment distance property values. Changing this value affects other settings that use angles, for example Standard Deviations.

Angle

Specifies the type of angle units to be used for the equipment angle property values.
When you change this value, other settings (for example Standard Deviations) that use angles are changed accordingly.

### Angle

Horizontal Collimation

Specifies the horizontal collimation correction applied to the FACE 1 and FACE 2 horizontal angles. The default value is 0.0 angle units.

Angle Type

Specifies the horizontal angle type. Select one of the following:

■ Right: measures the angle clockwise. (default)

■ Left: measures the angle counter-clockwise.

Vertical Collimation

Specifies the vertical collimation correction applied to the FACE 1 and FACE 2 vertical angles. Default value is 0.0 angle units.

Vertical Angle Type

Specifies the vertical angle type. Select one of the following:

**NOTE** A 90° Zenith angle equals a 0° Horizontal angle or a 270° Nadir angle. Also, an 89° Zenith angle equals a 1° Horizontal angle and a 269° Nadir angle.

■ Zenith: Select this value to measure the vertical angle between zenith and the object that is observed. (default)

■ Horizontal: Select this value to measure the vertical angle with respect to the horizon.

■ Nadir: Select this value to measure the vertical angle directly beneath the observer and directly opposite the zenith.

## Electronic Distance Meter (EDM)

Refractive Index

Specifies the refractive index for the carrier wavelength for the EDM, provided by the EDM manufacturer. Default value is 287.9600.

Wave Constant

Specifies the constant for the carrier wavelength for the EDM. Default value is 105.45.

Offset

Specifies the vertical distance between the scope center and the EDM center. Default value is 0.0000 mm.

Measuring Device

Specifies the type of device used for measuring distances. Select one of the following:

■ Scope: Select this value if the EDM is mounted above the scope; this type of EDM rotates radially about the horizontal axis as the scope is rotated.(default)

■ Non-scope: Select this value if the EDM is mounted on the scope; it is usually mounted on the standards and it does not rotate with the scope.

■ Other: Select this value if a device other than an EDM, such as steel tape or stadia rod, is used to measure distances.

## Prism

Tilting

Specifies the correction for EDM offset and prism offset. If this property is set to No, the following formulas are used to calculate the slope distance:
For a non-scope mounted EDM with non-tilting reflector:

$$SD = [(EO - PO) \times \cos VA] + \sqrt{OSD^2 - (PO - EO)^2 \times (\sin VA)^2}$$

For a scope mounted EDM with non-tilting reflector:

$$SD = \sqrt{OSD^2 - [EO - PO \times \sin VA]^2} - (PO \times \cos VA)$$

- EO: EDM offset

- PO: Prism offset

- OSD: Old slope distance

- SD: Slope distance

- VA: Vertical angle

**NOTE** Prism and EDM offset are not applied in tilting prism systems.

Prism Offset

Specifies a value for the prism offset (mm) to establish the vertical distance between the reflector (prism) and the target. Default value is 0.0000 distance units.

Prism Constant

Specifies the value for the prism constant (mm) to establish the horizontal distance between the effective prism and the actual prism placement. The value is added to all measured distances. Default value is 0.0000 distance units.

## Standard Deviations

Use these settings to specify the accuracy of the surveying equipment. These values are used to calculate the standard errors that appear in the least squares input file.

EDM Proportional Error (ppm)

Specifies the proportional error, in parts per-million, associated with the EDM. This value can also be determined from the manufacturers documentation for the EDM. Default value is 5.0000 ppm.

EDM Constant Error

Specifies the constant error, in millimeters, associated with the EDM. This value can also be determined from the manufacturers documentation for the EDM. Default value is 0.005 meters (5mm).

Centering Error

Specifies the centering error, in millimeters, involved with centering the instrument over a point. Default value is 0.0015 Meters (1.5 mm).

Target Alignment Error

Specifies the target alignment error, in millimeters, involved with aligning the instrument with the target. Default value is 0.0025 Meters (2.5 mm).

Pointing Error

Specifies the pointing error, in seconds of a degree, involved with pointing the instrument at the target. Default value is 0.0001 Degrees DMS (1.0 seconds).

Horizontal Circle

Specifies the horizontal circle error, in seconds of a degree, associated with reading a horizontal circle. Default value is 0.0015 Degrees DMS (1.5 seconds).

Direction Error

Specifies the direction error, in seconds of a degree, associated with the measurement of an azimuth or bearing. Default value is 0.0010 Degrees DMS (10 seconds).

Vertical Circle

Specifies the vertical circle error, in seconds of a degree, associated with reading a vertical circle. Default value is 0.00015 Degrees DMS (1.5 seconds).

Prism Height Error

Specifies the error, in millimeters, associated with measuring the prism height. Default value is 0.001 Meters (1.0 mm).

Theodolite Height Error

Specifies the error, in millimeters, associated with measuring the theodolite height. Default value is 0.001 Meters (1.0 mm).

# Translate Survey Database Dialog Box

Use this dialog box to specify the values needed to perform a simple translation on an open survey database.

This command is useful when you need to move the data in a survey database from an assumed location to a known location, subsequent to importing the survey data into AutoCAD Civil.

## Base Point Page (Translate Survey Database Dialog Box)

Specify the base point from which the survey network will be moved.

Number

Enter an existing survey point number in the survey database. The Name, Easting, Northing, Elevation, Description, Latitude, and Longitude property values are displayed.

**NOTE** The values for the Base Point properties are displayed in the current Survey database units.

Name

Displays the name of the base point.

Easting

Displays the Easting value for the base point.

Northing

Displays the Northing value for the base point.

Elevation

Displays the elevation value for the base point.

Description

Displays an optional description for the base point.

Longitude

Displays the Longitude for the base point. If the Survey database has been assigned a coordinate zone, the Longitude property displays the longitude of the easting value.

Latitude

Displays the Latitude value for the base point. If the Survey database has been assigned a coordinate zone, the Latitude property displays the latitude of the Northing value.

Pick In Drawing

In the drawing, select the northing and easting for the base point. If the drawing has been assigned a coordinate zone that is different from the assigned zone of the Survey Database, the coordinates of the point that you pick in the drawing are transformed to the Survey Database coordinates.

## Rotation Angle Page (Translate Survey Database Dialog Box)

Specify the rotation angle for the survey database.

Rotation Angle

Specify a positive or negative rotation angle.

---

**NOTE** The values for the Rotation Angle are displayed in the current Survey database units.

---

Pick In Drawing

Specify the start point of the existing direction, the second point, the start point of new direction, and a second point.

# Destination Point Page (Translate Survey Database Dialog Box)

Specify the destination point to which the survey database will be moved.

Easting

Specifies the easting value.

Northing

Specifies the northing value.

Longitude

Displays the Longitude for the base point. If the Survey database has been assigned a coordinate zone, the Longitude property displays the longitude of the easting value.

Latitude

Displays the Latitude value for the base point. If the Survey database has been assigned a coordinate zone, the Latitude property displays the latitude of the Northing value.

Elevation Change

Specifies the change in elevation. (optional)

Pick In Drawing

Specify the start point of the existing direction, the second point, the start point of new direction, and a second point.

# Summary Page (Translate Survey Database Dialog Box)

Displays a summary of the Base Point, Destination Point, and the Translation values

# Survey Network Style Dialog Box

Define survey network styles to control the appearance of network components such as markers and lines, and the display of components in the drawing.

# Information Tab (Survey Network Style Dialog Box)

Use this tab to change the survey network style name and description information, and to review details, such as when the style was most recently modified.

Name

Specifies a name for the current style. The style name identifies the style on the Settings tab in Toolspace and is listed in the Style Selection dialog box. Changes to a style name are reflected in the title bar of the Style dialog box.

Description

Specifies the description for the current style.

# Components Tab (Survey Network Style Dialog Box)

Use this tab to specify how survey network style components are displayed, such as component scaling, and symbols.

### Marker Styles

Use these settings to specify how markers are displayed. Markers are used to control the display of network points.

Marker Style For Known Control Points

   Specifies a marker style for known (fixed) control points in the survey network.

Marker Style For Unknown Control Points

   Specifies a marker style for the unknown control points in the survey network.
   Unknown points are the vertices of the network lines whose locations are determined from observations relative to the known Control Point locations. Typically, an unknown control point is an instrument setup. However, Sideshot observations made from unknown control points are recalculated when a network is updated from a network adjustment, such as a Traverse or Least Squares adjustment.

Marker Style For Non-Control Points

   Specifies a marker style for non control points in the survey network.

Marker Style For Sideshot Points

   Specifies a marker style for the sideshot points in the survey network.

Marker Style For Tolerance Error Points

   Specifies a marker style for the tolerance error points in the survey network.

### Error Ellipses

Error Ellipses Scale Factor

   Specifies a scale factor for displaying an error ellipse (page 2473) of a survey network.

# 3D Geometry Tab (Survey Network Style Dialog Box)

Use this tab to specify how the survey network style components display in 3D views.

### 3D Geometry

Network Display Mode

   Specifies the mode in which the network elevations are displayed in the drawing. Click the cell in the Value column and select one of the following from the list:

   ■ Use Network Elevations: Displays network components at the actual elevations.

   ■ Flatten Network To Elevation: Flattens the network component elevations to a specified value.

   ■ Exaggerate Network Elevations By Scale Factor: Scales network component elevations by a specified scale factor.

Flatten Network To Elevation

   Flattens the network component elevations to a specified value. This option is only active when you specify the value Flatten Network To Elevation as the Network Display Mode.

Exaggerate Network Elevations By Scale Factor

   Scales network component elevations by a specified scale factor. This option is only active when you specify the value Exaggerate Network Elevations By Scale Factor: as the Network Display Mode.

## Display Tab (Survey Network Style Dialog Box)

Use this tab to specify the survey network style component display in Plan (2D) and Model (3D) views. This is the standard AutoCAD Civil Display tab. For more information, see Display Tab (Style Dialog Box) (page 1801)

Survey Network Display components include:

- Known Control Points: a point symbol representing a known point in a survey network. All other unknown points are dependent on the location of a known point.

- Unknown Control Points: a point symbol representing a point in the survey network where the point location is calculated relative to an observation made to a known control point. An unknown control point may represent an instrument setup or an adjusted control point.

- Non-Control Points: a point symbol representing a point with its location determined from a reduced observation, for example a point created by the NE SS survey command. A non-control point is not connected to other survey observations and remains unaffected by network analysis, but it still a component of the survey network. A non-control point may be promoted to a control point if it is referenced by observations.

- Sideshot Points: a point symbol representing a sideshot point in the survey network with its location determined from an observation made from either a known or unknown point. In most cases a sideshot point is not connected to any other point in the network.

- Network Lines: a line representing observations that connect two control points.

- Direction Lines: a line representing observed direction data from either a known or unknown point in the survey network.

- Sideshot Lines: a line representing observations between a control point and a sideshot.

- Error Ellipses: this component represents the positional error of an unknown control point as a result of least squares analysis.

- Tolerance Error Points: this component represents an observed point where the observation(s) exceeds the tolerance error values specified in the Survey Database Settings.

- Tolerance Error Lines: this component represents the line of sight between the setup and the observed point that exceeds the tolerance error values that were specified in the Survey Database Settings.

## Summary Tab (Survey Network Style Dialog Box)

Use this tab to view all the survey network style properties. For more information about the properties in this tab, see Summary Tab (Style Dialog Box) (page 1803).

# Survey Figure Style Dialog Box

Use survey figure styles to control the way a figure and its components, such as points and lines, are displayed in the drawing.

# Information Tab (Survey Figure Style Dialog Box)

Use this tab to view or change general information for the survey figure style.

Name

Specifies the name of the current figure style.

Description
    Specifies an optional description for the current style.

Created By
    Displays the AutoCAD login name of the person who created the style.

Date Created
    Displays the date and time the style was created.

Last Modified By
    Displays the AutoCAD login name of the person who last modified the style.

Date Modified
    Displays the date and time the style was last modified.

# Plan and Model Tab (Survey Figure Style Dialog Box)

Use this tab to specify marker styles and marker placement options for each figure marker component. These figure marker components display on the Display tab of the figure style.

### Vertex Markers

Use these settings to specify the marker properties for figure Vertex Markers component. You can use the vertex markers component to create a marker style for the original data points from which the figure was created, for example the point located in field that defined the figure.

Marker Style

Specifies the marker style for the marker component. Click ⋯ to display the Figure Vertex Marker Style dialog to specify the marker style.

Align Markers With Figure
    Specifies if the markers are aligned normal to the figure segments. Click the cell in the Value Column and select to Yes, to override the specified marker style orientation properties.

### Midpoint Markers

Use these settings to specify the marker properties for the figure Midpoint Markers component. The midpoint markers component can be used to display a marker style for the midpoint of each figure segment.

Marker Style

Specifies the marker style for the marker component. Click ⋯ to display the Marker Style dialog to specify the marker style.

Align Markers With Figure
    Specifies if the markers are aligned normal to the figure segments. Click the cell in the Value column and select to Yes to override the specified marker style orientation properties.

### Endpoint Markers

Use these settings to specify the marker properties for the starting point and the ending point of the figure.

Start Point Marker Style
    Specifies the marker style for the start point of the figure.

End Point Marker Style
    Specifies the marker style for the end point of the figure.

Align Markers With Figure
    Specifies if the markers are aligned normal to the figure segments. Click the cell in the Value column and select to Yes, to override the specified marker style orientation properties.

**Additional Markers**

Use these settings to specify if additional markers and marker styles to use on figures. Markers are considered additional when they are not placed at a vertex, start, mid, or endpoint on the figure.

Marker Style

Specifies the marker style for the marker component. Click ![button] to display the Marker Style dialog to specify the marker style.

Additional Marker Placement Method

Specifies the placement of additional markers. Click the cell in the Value column and select one of the following from the list:

■ None: disables the placement of intermediate additional markers on the figure.

> **NOTE** if you specify this option, At Interval and Divide Figure By properties in the Additional Markers category are disabled.

■ At Interval: places additional markers at the specified interval.

■ Divide Figure: places markers on the figure segments determined by the Divide By property.

■ Continuous: specifies that additional markers are placed continuously along the figure. Each marker is placed outside the extents of the previous marker.

Interval

Specifies the interval if you set the Marker Placement Method to At Interval. Click the cell in the Value column and enter a number that is greater than zero.

Divide Figure By

Specifies the value to divide the figure by if you set the Marker Placement Method to Divide By. Click the cell in the Value column and enter a number that is greater than zero.

Place Marker At Figure Start Point

Specify Yes to have the Marker Placement Method always places an additional marker at the start point of the figure.

> **NOTE** This setting overrides the setting of None in the Marker Placement Method.

Place Marker At Figure End Point

Specify Yes to have the Marker Placement Method always places an additional marker at the end point of the figure.

> **NOTE** This setting overrides the setting of None in the Marker Placement Method.

Align Markers With Figure

Specifies if the markers are aligned normal to the figure segments. Click the cell in the Value column and select to Yes, to override the specified marker style orientation properties.

# 3D Geometry Tab (Survey Figure Style Dialog Box)

Use this tab to specify the 3D display of the survey figure style components.

Figure Display Mode

Specifies the display mode for figure elevations in the drawing. Click the cell in the Value column and select one of the following:

■ Use Figure Elevations: displays figure components at the actual elevations.

- Flatten Figure To Elevation: flattens figure component elevations to a specified value:

- Exaggerate Figure Elevations By Scale Factor: scales figure component elevations by a specified scale factor.

Flatten Figure To Elevation

This field is enabled if you specify Flatten Figure To Elevation as the Figure Display Mode. All figure component elevations are displayed at the specified elevation.

Exaggerate Figure Elevations By Scale Factor

This field is enabled if you specify Exaggerate Figure Elevations By Scale Factor as the Figure Display Mode. Click the cell in the Value column and enter a value. All figure component elevations are multiplied by the value you enter.

# Profile Tab (Survey Figure Style Dialog Box)

Use this tab to specify the marker styles for survey figures projected into profile views.

You can select different markers for the beginning, internal, and end vertices of a survey figure.

# Section Tab (Survey Figure Style Dialog Box)

Use this tab to specify the marker style for survey figures projected into section views.

This marker is used to indicate where a survey figure crosses the sample line displayed in the section view.

# Display Tab (Survey Figure Style Dialog Box)

Use this tab to specify the display styles for survey lines and markers in different contexts.

This is the standard AutoCAD Civil Display tab. For more information, see Display Tab (Style Dialog Box) (page 1801).

View Direction

Specifies the context for which you are setting the display:

- **Plan** – the printed drawing

- **Model** – the on-screen design window

- **Profile** – projection into a profile view

- **Section** – projection into a section view

Survey figure display components include:

- Figure Line

- Vertex Markers

- Midpoint Markers

- Endpoint Markers

- Additional Markers

## Summary Tab (Survey Figure Style Dialog Box)

Use this tab to view all the survey network style properties. For more information about the properties in this tab, see Summary Tab (Style Dialog Box) (page 1803).

# New Equipment Database Dialog Box

Use this dialog box to specify the name for a new equipment database.

Enter a unique name for the new equipment database.

---

**NOTE** A best practice is to create an equipment database per equipment manufacturer. For example, create one equipment database for Leica surveying instruments, and another for Trimble surveying instruments.

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**Related procedures:**

■ Survey Equipment Database (page 199)

# New Figure Prefix Database Dialog Box

Use this dialog box to specify the name for a new figure prefix database.

Enter a unique name for the new figure prefix database.

**Related procedures:**

■ Survey Figure Prefix Database (page 200)

# New Figure Prefix/Figure Prefix Properties Dialog Box

Use this dialog box to specify the properties for a new figure prefix.

Name

Specifies the name for the figure prefix. The prefix name is used to match on a figure name at figure creation time, for example, when importing a field book file.

Breakline

Specifies that the matched figures can be used as breaklines (when creating surface breaklines from figures).

Lot Line

Specifies that the figure behaves as a parcel segment. When this property is selected and the figure is inserted into drawing, the figure behaves as a parcel segment in the site that it is placed on. Figures with this property may create parcels and can be labeled as parcel segments.

Layer

Specifies the layer that the figure object is placed on when inserting figures into a drawing. When there is a prefix match, the following occurs:

■ If the specified layer exists in the drawing, the figure is placed on that layer.

■ If the specified layer does not exist in the drawing, the layer is created and the figure is placed on the specified layer.

Style

Specifies the figure style that is assigned to the figure when the figure is inserted into a drawing. When there is a prefix match, the following occurs:

■ If the specified style name exists in the drawing, the style is used.

■ If the specified style does not exist in the drawing, a new style with the specified name is created.

Site

Specifies the site name that is assigned to the figure if a figure prefix match occurs. The figure is placed in this site when it is inserted into a drawing. If the site name does not exist in the current drawing it is created.

**Related procedures:**

■ Survey Figure Prefix Database (page 200)

# Survey Panorama Vistas

Use the Survey Panorama vistas to display and edit collections of survey data, in a grid.

The Panorama displays many types of data. Each different type of data is displayed using a vista that is specific to that data type. To conserve screen space if more than one vista is active, the Panorama displays a tab for each one.

Click a tab to bring a vista to the front.

You can dock and move the Panorama, and you can use Auto-hide to reduce the amount of screen space it requires when you are not using it. For more information, see The Panorama Window (page 102).

## Control Points Editor

Use this dialog box to edit existing control points or create new ones.

Point Number

Specifies the point number.

Name

Specifies the point name. Optional.

Northing

Specifies the point northing.

**NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value entered in this field.

Easting

Specifies the point easting.

**NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value entered in this field.

Elevation

Specifies the point elevation.

Description

Specifies the point description.

**NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Latitude
   Specifies the point latitude. The Northing property value is calculated from the value entered in this field.

Longitude
   Specifies the point longitude. The Easting property value is calculated from the value entered in this field.

**Related procedures:**

■ The Panorama Window (page 102)

■ Control Points (page 253)

## Non-Control Points Editor

Use this dialog box to edit existing non-control points or create new ones.

Point Number
   Specifies the point number.

Name
   Specifies the point name.

Northing
   Specifies the point northing.

   **NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value entered in this field.

Easting
   Specifies the point easting.

   **NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value entered in this field.

Elevation
   Specifies the point elevation.

Description
   Specifies the point description.

**NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Latitude
   Specifies the point latitude. The Northing property value is calculated from the value entered in this field.

Longitude
   Specifies the point longitude. The Easting property value is calculated from the value entered in this field.

**Related procedures:**

■ The Panorama Window (page 102)

■ Non-Control Points (page 256)

# Setups Editor

Use this dialog box to view and edit survey setups.

**Station Point**

Displays the station point number.

**Backsight Point**

Displays the backsight point number.

**Backsight Direction**

Displays the direction of the backsight in the specified units.

**Backsight Orientation**

Specifies the orientation of the horizontal circle when backsighting the point. This is an optional value, and is assumed to be 0.00 unless you specify otherwise.

**Backsight Face1**

Specifies the direct angle reference on the backsight for face 1 (page 2473) angles.

**Backsight Face2**

Specifies the direct angle reference on the backsight for face 2 (page 2473) angles.

**Instrument Height**

Specifies the measured instrument (theodolite) height. This is typically measured from the center of the theodolite optics to the setup point on the ground.

**Instrument Elevation**

Specifies or displays the instrument elevation. If the setup point has an elevation, this field is read-only and displays the calculated instrument elevation (setup point elevation plus the instrument height). If the setup point has no elevation, then this column can be used to specify the instrument elevation.

**Northing**

Displays the northing value for the setup.

**Easting**

Displays the easting value for the setup.

**Elevation**

Specifies the elevation of the setup.

**Related procedures:**

- The Panorama Window (page 102)
- Setups (page 259)

# Directions Editor

Use this dialog box to view and edit survey directions.

**From Point**

Displays the originating control point.

**To Point**

Displays the destination reference point for the direction.

Direction

Specifies the direction value. The unit type is determined by the direction type setting. For more information, see Database Settings (page 216)

Direction Type

Specifies the direction type. Either an azimuth (page 2467) or a bearing (page 2468).

**Related procedures:**

■ The Panorama Window (page 102)

■ Directions (page 283)

## Observations Editor

Use this dialog box to view and edit survey observations.

Point Number

Specifies the point number.
When creating a new point and automatic point numbering is off, you must specify the point number. If automatic point numbering is on, then the points are automatically numbered.

Name

Specifies an optional name for the point.

Angle

Specifies the measured horizontal angle. It is assumed to be clockwise (right). Use a negative number (-) to turn counter-clockwise (left). Expressed in the current units.

Angle Type

Specifies the angle type to locate the point. Either angle (page 2467), deflection angle (page 2472), face 1 (page 2473), face 2 (page 2473), or bearing (page 2468).

Distance

Specifies the distance from the instrument point to the point being located. It is measured in the current units.

Distance Type

Specifies the distance type to locate the point. Specify slope (page 2482), horizontal, or None.

Vertical

Specifies the vertical angle or distance depending on the Vertical Type.

Vertical Type

Specifies the method of measuring the vertical. Either Vertical Angle or Vertical Distance.

Target Height

Specifies the vertical distance between the ground elevation and the focal point of the prism or stadia.

Target Type

Specifies the target type. Either prism (page 2480), stadia (page 2483), or None.

Scale Factor

Specifies the scale factor that affects how ground measurements are translated into grid measurements.The horizontal distances are then multiplied by this factor when the Scale Factor correction is checked in the Measurement Corrections property in the Survey Database Settings dialog.

Northing

    Displays the calculated northing for the point.

Easting

    Displays the calculated easting for the point.

Elevation

    Displays the calculated elevation for the point.

**Related procedures:**

- The Panorama Window (page 102)
- Observations (page 263)

## Survey Points Editor

Use this dialog box to view and edit all survey points.

The point icon in the first column indicates the point type:

- ⬖ Control point.
- ⬖ A control point derived from an adjustment, such Traverse or Least Squares analysis.
- ⬖ Non-control point.
- ⬖ Setup.
- ⬖ Observation.

Number

    Displays the point number.

Name

    Specifies the point name.

Easting

    Displays the point easting.

> **NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value in this field.

Northing

    Displays the point northing.

> **NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value in this field.

Elevation

    Displays the point elevation.

Description

    Specifies the point description.

> **NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Longitude

Displays the point longitude, if applicable.

Latitude

Displays the point latitude, if applicable.

**Related procedures:**

■ The Panorama Window (page 102)

# Figure Prefixes Editor

Use this dialog box to view and edit the properties of a figure prefix.

Name

Specifies the name for the figure prefix. The prefix name is used to match on a figure name at figure creation-time, for example when importing a field book file.

Breakline

Specifies that the matched figures can be used as breaklines (when creating surface breaklines from figures).

Lot Line

Specifies that the figure behaves as a parcel segment. When this property is selected and the figure is inserted into drawing, the figure behaves as a parcel segment in the site that it is placed on. Figures with this property may create parcels and can be labeled as parcel segments.

Layer

Specifies the layer that the figure object is placed on when inserting figures into a drawing. When there is a prefix match, the following occurs:

■ If the specified layer exists in the drawing, the figure is placed on that layer.

■ If the specified layer does not exist in the drawing, the layer is created and the figure is placed on the specified layer.

Style

Specifies the figure style that is assigned to the figure when the figure is inserted into a drawing. When there is a prefix match, the following occurs:

■ If the specified style name exists in the drawing, the style is used.

■ If the specified style does not exist in the drawing, it is created with the specified name.

Site

Specifies the site name that is assigned to the figure if a figure prefix match occurs. The figure is placed in this site when it is inserted into a drawing. If the site name does not exist in the current drawing it is created.

**Related procedures:**

■ Survey Figure Prefix Database (page 200)

■ The Panorama Window (page 102)

# Figures Editor

Use this dialog box to view and edit survey figures.

Name

Specifies the name of the figure. The figure name can be composed of characters that are number, alpha, or alpha-numeric.

Breakline

Specifies that the figure is used as a breakline (when creating surface breaklines from figures).

Lot Line

Specifies that the figure behaves as a parcel segment. When this property is selected and the figure is inserted into drawing, the figure behaves as a parcel segment in the site that it is placed on. Figures with this property may create parcels and can be labeled as parcel segments.

Layer

Specifies the layer that the figure object is placed on when inserting figures into a drawing.

Style

Specifies the style that the figure object uses when inserting figures into a drawing.

Site

Specifies the site for the figure. When inserting a figure into the current drawing:

■ If this site does not exist in the current drawing, it is created.

■ The figure is placed on this site. If the Lot Line property is checked for the figure, a parcel segment is created.

Vertices

Displays the number of vertices in the figure.

Length

Displays the length of the figure.

Area

Displays the area of the figure.

**Related procedures:**

■ The Panorama Window (page 102)

■ Survey Figures (page 321)

## Figure Groups Editor

Use this dialog box to select and edit a survey figures within a figure group.

Name

Specifies the name of the Figure Group.

**Related procedures:**

■ The Panorama Window (page 102)

■ Using Groups to Manage Survey Data (page 248)

## Network Groups Editor

Use this dialog box to select and edit a survey network from within a survey network group.

**LandXML and User-Defined**

> **NOTE** If you are using the Extended Properties feature, relevant LandXML and User-defined properties for each group are displayed in the panorama. To edit a group, select the group in the panorama and make edits in the Extended Data panel (left pane). Click  to open or close the extended data panel.

Name
   Specifies the name of the Network Group.

Description
   Specifies an optional description.

**Related procedures:**

■ The Panorama Window (page 102)

■ Using Groups to Manage Survey Data (page 248)

## Networks Editor

Use this dialog box to edit a survey network.

### LandXML and User-Defined

> **NOTE** If you are using the Extended Properties feature, relevant LandXML and User-defined properties for each Network are displayed in the panorama. To edit a network, select the network in the panorama and make edits in the Extended Data panel (left pane). Click  to open or close the extended data panel.

Name
   Specifies the Network name.

Description
   Specifies an optional description.

**Related procedures:**

■ The Panorama Window (page 102)

■ Survey Networks (page 201)

## Survey Point Groups Editor

Use this dialog box to select and edit a survey point within a survey point group.

### LandXML and User-Defined

> **NOTE** If you are using the Extended Properties feature, relevant LandXML and User-defined properties for each group are displayed in the panorama. To edit a group, select the group in the panorama and make edits in the Extended Data panel (left pane). Click  to open or close the extended data panel.

Name
   Specifies the name of the Survey Point Group.

Description
   Specifies an optional description.

# Figure Display

Use this dialog box to display Inverse or Mapcheck information for a figure.

# Traverse Editor

Use this dialog box to edit the observations of an existing named traverse or to enter traverse observations for a new empty traverse.

The left pane of the Traverse Editor displays the following properties for the selected setup within the current station point:

Station Point
   Displays the station point number.

Backsight Point
   Displays the backsight number.

Backsight Direction
   Specifies the backsight direction.

Backsight Orientation
   Specifies the orientation of the horizontal circle when backsighting the point.

Backsight Face1
   Specifies the direct angle reference on the backsight for face1 angles.

Backsight Face2
   Specifies the direct angle reference on the backsight for face2 angles.

Instrument Height
   Specifies the measured instrument height.

Instrument Elevation
   Displays the instrument elevation.

Northing
   Displays the northing value.

Easting
   Displays the easting value.

Elevation
   Displays the elevation.

**Latitude**

Displays the latitude.

**Longitude**

Displays the longitude.

In the right pane of the Traverse Editor you can enter observations for each Setup and specify the following;

**Name**

Specifies a traverse station point name.

**Angle**

Specifies the angle using the current Survey database units.

**Angle Type**

Specify one of the following:

- Angle

- Deflection

- Face1 Angle

- Face2 Angle

- Azimuth

- Bearing-NE

- Bearing-SE

- Bearing-SW

- Bearing-NW

- None

**Distance**

Specify the distance in the current Survey database units.

**Distance Type**

Specify one of the following:

- Slope

- Horizontal

- None

**Vertical**

Specifies the vertical type.

**Vertical Type**

Specify one of the following:

- Vertical Angle

- Vertical Distance

- None

**Target Height**

Specifies the target height.

Target Type

Specify one of the following:

■ Stadia

■ Prism

■ None

Scale Factor

Specifies the scale factor.

Description

Specifies a description.

Easting

Displays the easting.

Northing

Displays the northing.

Elevation

Displays the elevation.

Longitude

Displays the latitude.

Latitude

Displays the longitude.

**Related procedures:**

■ The Panorama Window (page 102)

## Traverses Editor

Use this vista to edit and view information for multiple traverses.

Name

Specifies the name for the traverse.

Description

Specifies a description for the traverse.

Initial Station

Specifies the starting setup point ID in the traverse.

Initial Backsight

Specifies the initial backsight. The initial backsight point number must not be 0. The initial backsight must be to either an existing survey point, or there must be a direction observation from the initial station point to the initial backsight point. In cases where there is a direction observation, the backsight point does not have to exist as a survey point.

Stations

Specifies the points that are included in the traverse. Separate a range of points with a hyphen (-), and separate individual points with a comma (,). For example: 1-5,8,10,12-15.

Final Foresight

Specifies the final foresight point ID. This establishes the final foresight station point in the traverse definition, onto which the closing angle was turned.

**Related procedures:**

■ Traverses (page 373)

■ The Panorama Window (page 102)

# Import / Re-import Field Book Dialog Box

Use this dialog box to edit user and survey database settings associated with a field book import operation.

The current user and survey database setting defaults are displayed when the dialog is initialized. The settings here are not saved back to the user and survey database settings and are only applicable for the field book import operation.

Field Book Filename (.FBK)

Specifies the field book (*.fbk*) to import. Click ![browse icon] to browse to a location.

Current Equipment Database

Specifies the equipment database to use. Select the equipment database from the drop-down list.

**NOTE** The list of available databases is determined by the databases listed in the Equipment Databases collection in the Toolspace Survey tab.

Current Equipment

Specifies the current equipment to use. Click the equipment name from the drop-down list.
The current equipment sets the values associated with a specific surveying instrument, such as the standard deviations associated with the measuring capabilities of the equipment. This information is used in various calculations including least squares.

Current Figure Prefix Database

Specifies the current figure prefix database. Click the figure prefix database from the drop-down list. Figure prefixes enable you to determine the layer that a figure is drawn on by creating a prefix for a group of figure names. All figures with a specific prefix are placed on a specific layer

**NOTE** The list of available databases is determined from the databases listed in the Figure Prefix Databases collection on the Toolspace Survey tab.

Show Interactive Graphics

Specifies whether to display interactive graphics during the import of survey data. Use Interactive Graphics settings to control the display of individual graphics components.

Process Linework During Import

Specify Yes to process linework during the import.

Current Linework Code Set

Specifies the default linework code set to use when linework processing occurs.

Process Linework Sequence

Specifies the order in which survey point descriptions are processed to determine linework connectivity. Select one of the following:

■ By Import Order - processes points in the order which they are imported. Point names are always processed by import order.

- By Point Number - processes points sequentially by point number (ascending order only).

Import Event Name

Specifies a file name for the import event. The name of the file you are importing is displayed as the default name.

Import Event Description

Optionally, enter a description of the import event.

Assign Offset To Point Identifiers

Specifies that an offset is applied to the point identifier before the points are imported into the survey database.

Point Identifier Offset

Specifies the offset value that is applied to the point identifier. For example, if the offset value is 1000, the point number 101 is renumbered to 1101, or point name KC101 is renamed as KC101.1000.

Insert Network Object

Specifies whether a network object is created in the drawing upon completion of the import command.

Insert Figure Objects

Specifies whether figure objects are created in the drawing upon completion of the import command.

Insert Survey Points

Specifies whether AutoCAD Civil points are created in the drawing upon completion of the import command.

Display Tolerance Errors In Event Viewer

Select Yes to display tolerance errors in the Event Viewer when you import field book files. For more information, see Reporting Tolerance Errors (page 211).

**Related procedures:**

- Survey Database Settings Dialog Box (page 2378)
- Survey User Settings Dialog Box (page 2374)
- Field Book Files (page 240)

# Least Squares Analysis Dialog Box

Use this dialog box specify Least Squares settings. The settings default to the setting in the Survey Database Settings for Least Squares.

**Input**

Create Input File

Specifies that the Input file name is created, or overwritten if it already exists, in the *<Working folder>\<Project>\Survey\<Network>* folder.

Input File Name

Specifies name of the least squares input file. If the Create Input File property is checked, the input file name (without the **\***.*lsi* extension) can be entered in the combo box, or an existing input file can be selected from the list. If an existing input file is selected it is overwritten with new content.

Network Adjustment Type

Changes the default network adjustment type of the least analysis performed on the survey network. Specify one of the following:

■ 2-Dimensional: The input file contains content to perform a 2-dimensional least squares analysis.

■ 3-Dimensional: The input file contains content to perform a 3-dimensional least squares analysis.

**NOTE** This property is enabled if the Create Input File property is checked.

### Procedure

Maximum Number of Iterations

Specifies the number of iterations of least squares analysis that is performed on the network.

Coordinate Convergence

Specifies the default coordinate convergence of the least squares analysis performed on the survey network.

Confidence Level

Specifies the confidence level of the least squares analysis performed on the survey network.

Perform Blunder Detection

Specify Yes to perform Blunder Detection on the least squares analysis performed on the survey network.

Update Survey Database

Specify Yes to update the Survey Database with the adjusted coordinates from the analysis.

# Traverse Analysis Dialog Box

Use this dialog box to enter settings for traverse analysis. Initially, the Traverse Analysis settings you enter in the Survey Database settings are displayed as the default settings.

### Traverse Analysis Property

Do Traverse Analysis

Select Yes to perform a mathematical analysis based on the method you specify as the Horizontal Adjustment Method.

Do Angle Balance

Select Yes to have the angular error evenly distributed throughout the traverse.

Horizontal Adjustment Method

Specify one of the following traverse analysis methods:

■ Compass Rule: A method of corrections where the closing errors are assumed to be as much due to errors in observed angles as by the errors in measured distances. The closing errors in latitude and departure are distributed according to the ratio of the length of the line to the total length of the traverse

■ Transit Rule: A method of balancing a traverse where the closing errors are assumed to be caused less by the errors in the observed angles than by the errors in measured distance. Corrections are distributed according to the ratio of the latitude and departure of each leg of the traverse to the sums of the latitude and departures of the entire traverse.

■ Crandall Rule: A method of balancing a traverse where all the angular error is distributed throughout the traverse and all adjustments to the traverse are due to modifying the traverse distances. The modification distance made to each leg is such that the sum of the squares is a minimum. Corrections corresponding to the closing errors assume that the closing errors are random and normally distributed, and that all the angular error has been adjusted prior to the adjustment routine.

- Least Squares: A method of balancing a traverse. The squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method weights the individual measurements according to the specifications set in the Equipment correction settings to determine the source of error. You can adjust the data for an individual traverse loop, or a traverse network, if located from multiple observations.

Vertical Adjustment Method

Specify one of the following vertical adjustment methods:

- None: No vertical adjustment performed

- Length Weighted Distribution: Vertical adjustment distributes the vertical closing error to each line at the same ratio as the length of the line to the total length of the traverse (similar to the Compass rule).

- Equal Distribution: Vertical adjustment distributes the vertical closing error equally to each of the traverse stations.

- Least Squares: A method of distributing the vertical error throughout a traverse. The squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method weights the individual measurements according to the specifications set in the Equipment correction settings to determine the source of error. You can adjust the data for an individual traverse loop, or a traverse network, if located from multiple observations.

**NOTE** The Least squares vertical adjustment method is only available if Least squares has been selected as the horizontal adjustment method. When this method is selected, a 3D least squares adjustment is performed. If this method is not selected a 2D least squares adjustment is performed.

Horizontal Closure Limit 1:X

Specifies the minimum allowable error of closure in the horizontal direction, for example a value of 15000.00 is 1 part in 15000.

Vertical Closure Limit 1:X

Specifies the minimum allowable error of closure in the vertical direction, for example a value of 15000.00 is 1 part in 15000.

Angle Error Per Set

Specifics the maximum angular error per traverse setup.

Update Survey Database

Specify Yes to update the Survey database with the adjusted coordinates.

**NOTE** During the analysis, if the specified limits for the horizontal, vertical, or angle limits are exceeded, a warning message box is displayed. Select Yes to update the database or No if you do not want the database updated.

# Preview Settings Dialog Box

Use this dialog box to specify the preview settings for networks, figures, and setups.

Previews are displayed in the Toolspace item graphical view. For more information, see The Toolspace Item View (page 82).

Select the check box for the components that you want to display in the preview.

# Survey Network Properties Dialog Box

Specify a name and styles for the network components, 3D Geometry, and Display options.

Name

Displays the name of the current network.

Description

Displays the description of the current network.

Object Style

Specifies the style used to display the network. Select a style from the list or use the standard selection tools. For more information about the standard selection tools, see Select Style Dialog Box (page 1805).

Show Tooltips

Controls whether tooltips are displayed for the object in the drawing (does not affect tooltips over toolbar icons).

## LandXML

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

## User-defined

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

# Create Figure From Object Dialog Box

Use this dialog box to create a figure from an AutoCAD Civil or AutoCAD object.

The following object types can be selected in the drawing to define a survey figure:

- 2D Line

- 3D Line

- 2D Polyline

- 3D Polyline

- Feature Line

- Lot Line

- Parcel (select the parcel area label)

Name

Displays the name of the current figure. The name entered here is checked for a match in the current figure prefix database. If the name matches to a figure prefix the properties in the figure prefix is assigned to the new figure. If the name does not match a figure prefix, then the figure is assigned the default figure style from the User Settings.

Current Figure Prefix Database

Displays the default figure prefix database that is assigned in the User Settings. This is the figure prefix database that is used to match against the Name property value.

Associate Survey Points To Vertices

Specifies to associate survey points to vertices. When this property is checked, for each figure vertex:

- The X-Y coordinates are checked for a match with an X-Y location of an existing survey point in the survey database.

- If the vertex X-Y coordinates match a survey point, then figure vertex references that survey point.

**Related procedures:**

- Creating Figures (page 321)

# Create Breaklines Dialog Box

Use this dialog box to add breaklines to a surface from existing figures.

Select Surface

Specifies the surface for which you want to add the breakline data. Click the surface in the drop-down list or click ⬦ to select a surface in the drawing.

Name

Displays the name of the figure.

Breakline

Specifies if the figure is to be used as breakline data in the surface. Only figures that have their Breakline property checked are displayed in this list. Select the check box to create breaklines from the figure.

**Related procedures:**

- Creating Breaklines from Figures (page 324)

# Survey Figure Properties Dialog Box

Specify or change the figure properties such as name, layer, style, and the site the figure is associated with.

Name

Specifies the name of the figure. The figure name can be composed of characters that are number, alpha, or alpha-numeric.

Breakline

Specifies that the figure is used as a breakline (when creating surface breaklines from figures).

Lot Line

Specifies that the figure behaves as a parcel segment. When this property is selected and the figure is inserted into drawing, the figure behaves as a parcel segment in the site that it is placed on. Figures with this property may create parcels and can be labeled as parcel segments.

Layer

Specifies the layer that the figure object is placed on when inserting figures into a drawing.

Style

Specifies the style that the figure object uses when inserting figures into a drawing.

Site

Specifies the site for the figure if Lot Lines property is enabled. When inserting a figure into the current drawing, if the site does not exist in the current drawing, it is created. The figure is placed on the specified site

Vertices
  Displays the number of vertices in the figure.

Length
  Displays the length of the figure.

Area
  Displays the area of the figure.

Network
  Displays the name of the network that the figure is associated with.

Import Event
  Displays the name of the import event that the figure is associated with.

Auto-generated
  Displays Yes or No to indicate whether the figure was automatically generated.

**LandXML**

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

**User-defined**

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

**Related procedures:**

■ Survey Objects (page 194)

# New Local Survey Database Dialog Box

Use this dialog box to specify the name for a new survey database.

Local survey databases are not stored in Vault. You can subsequently convert the database to a Vault project.

Name
  Specify the name for the new survey database.

**Related procedures:**

■ Survey Database (page 196)

# New Survey Network Dialog Box/Survey Network Properties Dialog

Use this dialog box to specify the name for a new survey network.

New Survey Network
  Specify the name for the new survey network.

**LandXML**

If you are using the extended properties feature, the LandXML elements as defined in the .sdx_def file are displayed. Click the cell in the Value column to assign an attribute.

**User-defined**

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

**Related procedures:**

■ Survey Networks (page 201)

## Batch File Dialog Box

Use this dialog box to specify the name for a new batch file.

Enter Name For Batch File
Specify the name for the new batch file.

**Related procedures:**

■ Batch Files (page 318)

## Output File Dialog Box

Use this dialog box to specify the name for a new output file.

Enter Name For Output File
Specify the name for the new output file.

**Related procedures:**

■ Outputting Survey Files (page 388)

## Export Field Book Dialog Box

Use this dialog box to edit settings associated with exporting AutoCAD Civil survey data to a field book.

The settings here are not saved back to the user settings and are only applicable for the field book export operation.

Field Book Filename (.FBK)

Specifies the name and location of the field book. Click  to specify a different name and location.
For information on the conditions and syntax that is added to the field book, see Setting Export Defaults (page 213).

Export Figures With Network
Select Yes to export the figures in the database when you use the Export Field Book command.
The figure type determines what is added to the field book. For information on the conditions and syntax that is added to the field book, see Setting Export Defaults (page 213).

Export Point Identifiers
Specifies if point IDs are used to define figures when figures are exported to a field book.
For information on the conditions and syntax that is added to the field book, see Setting Export Defaults (page 213).

Export Point Data
This check box is active when the Export Point Identifiers property is not set to None. Select Yes if you want each figure vertex that references a point identifier to output the following to the field book file prior to the section defining the figures: NE SS <point ID> <north> <east> <elevation> <description>

# New Control Point/Control Point Properties Dialog Box

Use this dialog box to specify or edit the properties of a control point.

If you are editing the properties of a control point using the Survey tab item view, for more information on the functions available, see Survey Tab Item View (page 2383).

Point Number

Specifies the point number.

Name

Specifies the point name. Optional.

Easting

Specifies the point easting. Required.

**NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value entered in this field.

Northing

Specifies the point northing. Required.

**NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value entered in this field.

Elevation

Specifies the point elevation. Optional.

Description

Specifies the point description. Optional.

**NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Longitude

Specifies the point longitude. The Easting property value is calculated from the value entered in this field.

Latitude

Specifies the point latitude. The Northing property value is calculated from the value entered in this field.

Network

Displays the name of the network that first observed the point.

Import Event

Displays the name of the import event if the survey point was created by an import command.

Original Number

Displays the original survey point number if a point identifier offset was applied during the import command.

Original Name

Displays the original survey point name if the survey point was renamed during the import command.

**LandXML**

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

**User-defined**

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

**Related procedures:**

■ Control Points (page 253)

# New Non-Control Point/Non-Control Point Properties Dialog Box

Use this dialog box to specify or edit the properties of a non-control point.

If you are editing the properties of a non-control point using the Survey tab item view, for more information on the functions available, see Survey Tab Item View (page 2383).

Point Number
  Specifies the point number.

Name
  Specifies the point name. Optional.

Northing
  Specifies the point northing. Required.

  **NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value entered in this field.

Easting
  Specifies the point easting. Required.

  **NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value entered in this field.

Elevation
  Specifies the point elevation. Optional.

Description
  Specifies the point description. Optional.

**NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Latitude
  Specifies the point latitude. The Northing property value is calculated from the value entered in this field.

Longitude
  Specifies the point longitude. The Easting property value is calculated from the value entered in this field.

Network
  Displays the name of the network that first observed the point.

Import Event
  Displays the name of the import event if the survey point was created by an import command.

Original Number
  Displays the original survey point number if a point identifier offset was applied during the import command.

Original Name

Displays the original survey point name if the survey point was renamed during the import command.

**LandXML**

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

**User-defined**

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

**Related procedures:**

■ Non-Control Points (page 256)

# Survey Point Properties Dialog Box

Use this dialog box to view or edit the properties of a survey point.

Survey points can be control points, non-control points, setups, or observations.

If you are editing the properties of a point using the Survey tab item view, for more information on the functions available, see Survey Tab Item View (page 2383).

Number

Displays the point number.

Name

Specifies the point name.

Easting

Displays the point easting.

**NOTE** If the survey database has been assigned a coordinate system zone, the Longitude property value is calculated from the value in this field.

Northing

Displays the point northing.

**NOTE** If the survey database has been assigned a coordinate system zone, the Latitude property value is calculated from the value in this field.

Elevation

Displays the point elevation.

Description

Specifies the point description.

**NOTE** Latitude and Longitude values are only displayed when the survey database has an assigned coordinate system zone.

Longitude

Displays the point longitude. The Easting property value is calculated from the value entered in this field.

Latitude

Displays the point latitude. The Northing property value is calculated from the value in this field.

Network

Displays the name of the network that first observed the point.

Import Event

Displays the name of the import event if the survey point was created by an import command.

Original Number

Displays the original survey point number if a point identifier offset was applied during the import command.

Original Name

Displays the original survey point name if the survey point was renamed during the import command.

**LandXML**

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

**User-defined**

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

# New Direction/Direction Properties Dialog Box

Use this dialog box to specify or edit the properties of a direction.

For more information on the functions available if you are editing the properties of a direction using the Survey tab item view, see Survey Tab Item View (page 2383).

From Point

Specifies the originating control point.

To Point

Specifies the destination point reference for the direction.

Direction

Specifies the direction value. The unit type is determined by the direction type setting. For more information, see Database Settings (page 216).

Direction Type

Specifies the direction type. Either an azimuth (page 2467) or a bearing (page 2468).

**Related procedures:**

■ Directions (page 283)

# New Setup/Setup Properties Dialog Box

Use this dialog box to specify or edit the properties of a setup.

If you are editing the properties of a setup using the Survey tab item view, for more information on the functions available, see Survey Tab Item View (page 2383).

Station Point

Specifies the station point number for the setup.

NOTE If you enter a non-existent station point, you are prompted to create a point and the New Control Point dialog box is displayed from which you can create a new station point.

Backsight Point

Specifies the backsight point for the setup. When this column contains a valid entry, the Backsight Direction column displays the calculated backsight direction value and is read-only.

Backsight Direction

Specifies the backsight direction. This field is read-only if the Backsight Point column contains a valid point number.

Backsight Orientation

Specifies the backsight orientation angle for the setup. This is the reference angle for the all angle observations except Face 1 and Face 2 angles.

Backsight Face 1

Specifies the direct angle reference on the backsight for Face 1 angles.

Backsight Face 2

Specifies the direct angle reference on the backsight for Face 2 angles.

Instrument Height

Specifies the measured instrument (theodolite) height. This is typically measured from the center of the theodolite optics to the setup point on the ground.

Instrument Elevation

Specifies the instrument elevation.

NOTE If the setup point has an elevation, this field is read-only and displays the calculated instrument elevation (setup point elevation plus the instrument height). If the setup point has no elevation, then this column can be used to specify the instrument elevation

Easting

Displays the point easting.

Northing

Displays the point northing.

Longitude

Displays the point longitude.

Latitude

Displays the point latitude.

Setup Equipment Properties

Displays the name of the equipment properties for the selected setup. To edit the Setup Equipment Properties, click ⌹ to display the Setup Equipment Properties dialog box.

**Related procedures:**

■  Setups (page 259)


# Edit Setups That Observe Dialog Box

Use this dialog box to specify the point identifier for the observation that you want to find.

All setups that observe the point are displayed in the Setups Editor vista.

**Related procedures:**

■  Setups (page 259)

# New Traverse/Traverse Properties

Use this dialog box to specify or edit the properties of a traverse.

If you are editing the properties of a traverse using the Survey tab item view, for more information on the functions available, see Survey Tab Item View (page 2383).

Name
Specifies the name of the traverse. The traverse name can be composed of characters that are number, alpha, or alpha-numeric.

Description
Specifies a description for the traverse.

Initial Station
Specifies the starting station point.

Initial Backsight
Specifies the initial backsight point.

Stations
Specifies the setups as a comma delimited list.

Final Foresight
Specifies the final foresight point.

**Related procedures:**

■ Traverses (page 373)


# Figure Properties (Drawing Object)

Use this dialog box to specify a style for the figure drawing object.

Name
Specifies the name of the figure. The figure name can be composed of characters that are number, alpha, or alpha-numeric.

Style
Specifies the figure style.

**Related procedures:**

■ Survey Figures (page 321)


# Survey Command Window

Use this dialog box to enter survey commands directly using command line input or interactively using the menus.

The menus on this dialog box provide quick access to extended survey functionality including batch and output file usage, baseline, centerline, and intersection input, and to obtain point information.

Use the command input field to enter survey command language commands directly. For more information, see Survey Command Reference (page 391).

Main menu

Provides access to:

- [User settings](page 205)
- [Database settings](page 216)
- [Batch file functionality](page 318)
- [Output file functionality](page 388)
- [Baseline functions](page 285)
- [Centerline functions](page 289)
- [Intersection functions](page 304)
- [Point Information](page 376)
- Drawing zoom and pan functions

Output view

Displays all survey command output. All output is prefixed with the "!" character. Erroneous commands are recorded as comments. For example: !Duplicate point. (Point number ' 1 ' already exists).

Batch view

Displays the input history for the current Survey Command Window session. All commands echoed here are also written to the batch file (if turned on). Input errors are trapped and not recorded to the batch file.

Command line

Enter survey command language commands directly. The survey command language is the basic format for all survey data entry. After you become familiar with the survey command Language, you may find that it is an efficient way to access and query data. Commands entered at the survey command line use a specific syntax.

**Related procedures:**

- [Survey Command Language Commands](page 393)
- [Survey User Interface](page 193)

# Import / Re-import Survey LandXML Data Dialog Box

Use this dialog box to specify the import settings and the survey data to import from a LandXML file.

## Import Settings

Current Equipment Database

Specifies the current equipment database. Click the equipment database from the drop-down list. For more information, see [Survey Equipment Database](page 199).

NOTE  The list of available databases is determined by the databases listed in the Equipment Databases collection in the Toolspace Survey tab.

Current Equipment

Specifies the current equipment to use. Click the equipment name from the drop-down list.
The current equipment sets the values associated with a specific surveying instrument, such as the units used to measure angles and distances, and/or the standard deviations associated with the equipment's measuring capabilities. This information is used in various calculations including least squares.

Current Figure Prefix Database

Specifies the current figure prefix database. Click the figure prefix database from the drop-down list. Figure prefixes enable you to determine the layer that a figure is drawn on by creating a prefix for a group of figure names. All figures with a specific prefix are placed on a specific layer.

The list of available databases is determined from the databases listed in the Figure Prefix Databases collection on the Toolspace Survey tab.

Process Linework During Import

Click Yes to process line work during the import.

Current Linework Code Set

Specifies the linework code file to use when linework processing occurs.

Process Linework Sequence

Specifies the order in which survey point descriptions are processed to determine linework connectivity. Select one of the following:

■ By Import Order - processes points in the order which they are imported. Point names are always processed by import order.

■ By Point Number - processes points sequentially by point number (ascending order only).

Import Event Name

Specify a name for the import event. The name of the file you are importing is displayed by default.

Import Event Description

Optionally, enter a description of the import event.

Assign Offset To Point Identifiers

Specifies that an offset is applied to the point identifier before the points are imported into the survey database.

Point Identifier Offset

Specifies the offset value that is applied to the point identifier. For example, if the offset value is 1000, point number 101 is renumbered to 1101, or point name KC101 is renamed as KC101.1000.

Insert Network Object

Specifies whether to create a network object in the drawing.

Insert Figure Objects

Specifies whether to create figure objects in the drawing.

Insert Survey Points

Specifies that survey points are inserted into the drawing.

Display Tolerance Errors In Event Viewer

Select Yes to display tolerance errors in the Event Viewer when you import Survey LandXML files. For more information, see Reporting Tolerance Errors (page 211).

**LandXML Data tree**

Lists the LandXML header information for each data component that can be imported. Only the following LandXML components are displayed in the LandXML Data tree (if they exist in the selected LandXML file). Non-survey LandXML elements are not displayed.

Expand each of the collections to display all of the sub-components.

Select or clear the check boxes to the left of each data component name to filter the data that is imported into AutoCAD Civil. By default, all data components are selected for import.

File Name

Displays the path and file name of the selected LandXML file.

CoordinateSystem

If current Coordinate Zone property in the Survey Database Settings is set and this value is different, a coordinate transformation is attempted (the Survey Database Settings Distance property is also used to scale the coordinate values during the transformation).

Units

If the Units ➤ linearUnit setting is different from the Survey Database Settings ➤ Units Distance property, observation distance and target heights values are converted.
If the Units ➤ angularUnit settings is different from the Survey Database Settings ➤ Units Angle property, observation angle (horizontal, vertical, and direction) values are converted.

**NOTE** The internal units of the survey database are angle values in Radians and distance and coordinate values in Meters.

SurveyHeader

This element maps to a survey network in AutoCAD Civil. The Name attribute sets the name of new network in the survey database, or if a network of the same name exists in the survey database, it may be overwritten or appended to, depending on the LandXML Import Settings.

PlanFeatures

Maps to a survey figure. For each PlanFeature:

■ The PlanFeature Name attribute is the name assigned to the new figure.

■ If the PlanFeature name matches a figure prefix in the current figure prefix database, the figure prefix properties are assigned to the new or existing figure.

Parcels

A Parcel maps to a survey figure. For each Parcel:

■ The Parcel Name attribute is the name assigned to the new figure.

■ If the Parcel name matches a figure prefix in the current figure prefix database, the figure prefix properties are assigned to the figure, except for the Lot Line property.

■ The Lot Line property is always set to True for each Parcel imported into a figure. This overrides a figure prefix property if a figure prefix match occurs.

**NOTE** If the figure prefix does not have a Site name, the default Site name is used when the figure is inserted into a drawing.

CgPoints

A CgPoint point maps to a survey point. Each CgPoint is added to the Non-Control Points collection in the survey database.

**Related procedures:**

■ Importing Survey LandXML Data (page 244)

■ Importing Survey XML Data

# Export Survey LandXML Data Dialog Box

Use this dialog box to select the data in the export settings and survey data to export in LandXML format.

**Export Settings**

File Name

Specifies the file name to contain the exported LandXML survey data. Enter a path or click  to browse to a location.

Coordinate Zone

Click  to display the Select Coordinate Zone dialog box where you can specify the coordinate zone for the survey database data. The default zone is set to None.

Distance

Specifies the distance unit. Click the cell in Value column and select a distance unit from the list.

**NOTE** If you specify a Coordinate Zone, the distance unit is obtained from the zone and the Distance property field is not active.

Angle

Specifies the angle unit for the survey data. Click the cell in the Value column and select an angle type from the list.

Direction

Specifies the direction unit for the survey data. Click the cell in the Value column and select a direction type from the list.

Temperature

Specifies the temperature unit for the survey data when temperature corrections are applied to observations. Click the cell in the Value column and select a temperature type from the list.

Pressure

Specifies the atmospheric pressure unit for the survey data when pressure corrections are applied to observations. Click the cell in the Value column and select a pressure type from the list.

**Export Data**

Data tree

Lists the AutoCAD Civil survey data that you can export in LandXML format. Expand each of the collections to display all of the sub-components.
Select or clear the check box to the left of each feature name to filter the data that you want to export in LandXML format.

**NOTE** The check boxes have a tri-state display. If only some items are selected under a collection, the check box is dimmed  ; if all the items are selected, the check box is selected  ; if all the items below the collection are cleared, the check box for the collection is cleared  .

OK

Converts AutoCAD Civil objects to LandXML format and exports the LandXML data.

**Related procedures:**

■ Exporting Survey Data to LandXML (page 244)

# Astronomic Direction Calculator Dialog Box

Use this calculator to calculate astronomic directions from Sunshots and Starshots.

Calculation Type

Specifies the calculation types. Select one of the following from the type list:

- Sun shot calculation: Calculates the astronomic direction from solar observations by the hour angle method and uses a multiple foresight field method.

- Star shot calculation: Calculates the astronomic direction from Polaris or star observations by the hour angle method and uses a single foresight method.

## Observation Station Data

Station Point

Specifies the point ID for the station for which you want to calculate the direction.

Backsight Point

Specifies the backsight point ID for the station for which you want to calculate the direction.

Station Latitude

Specifies the Latitude of the station in the drawing's ambient settings format. Specify - for south.

Station Longitude

Specifies the Longitude of the station in the drawing's ambient settings format. Specify - for west.

UT1 Time

Specifies the UT1 time, which is determined by adding the correction (DUT) to the Coordinated Universal Time (UTC). UTC time is broadcast over WWV and other radio stations. You can also get DUT corrections via WWV; they are broadcast after the start of each minute.

## Ephemeris Data

GHA 00 Hours

Specifies the Greenwich Hour Angle (GHA) at zero (0) hours for the day on which the observations were made.

GHA 24 Hours

Specifies the GHA at 24 hours for the day the observations were made.

Declination 00 Hours

Specifies the declination at zero (0) hours on the day the observations were made.

Declination 24 Hours

Specifies the declination at 24 hours on the day the observations were made.

Sun Semi-diameter

Specifies the semi-diameter of the sun.
Enter zero (0) if you are sighting either the center or both the trailing and leading edges of the sun. The left edge is always the leading edge at latitudes greater than 23.5 degrees north and greater than 23.5 degrees south. If only the left edge is being sighted (left when facing the sun), then the semi- diameter is positive; likewise, the semi-diameter is negative when only the right edge is sighted.

## Observations



Creates a new observation set.

Deletes the selected observation set.

**Observation Sets**

Displays each observation set.

**Backsight Observation**

Specifies the backsight orientation.

**Sun Observation or Star Observation**

Specifies the sun or star observation.

**Stop Time**

Specifies the stop time for the observation.

**Observed Direction**

Calculated observed direction.

**Average Direction**

Calculated average direction.

**Mean Direction (Station To Backsight)**

Value calculated by the editor.

**Related procedures:**

■  Astronomic Direction Calculations (page 389)

# Manage Extended Properties Dialog Box

Use this dialog box to manage the extended property definitions in the Survey *.sdx* file.

 Expands All

Expands all items in the tree.

 Collapses All

Collapses all items in the tree.

 Export Settings

Exports the extended properties schema to a specified *.sdx* file. You specify the location of this *.sdx* file in the Database Defaults of Survey User Settings.

 Import Settings

Imports an extended properties definition schema into the existing *.sdx* file.

**NOTE** Existing extended property definitions that are referenced by an instance of a SurveyFeatureClass are not redefined during the import process. Existing extended property definitions that are not referenced are redefined by the definition in the selected *.sdx_def* file.

 Select All

Selects all category and property check boxes.

 Clear All

Clears all category and property check boxes.

 Create New

Displays the New Extended Property dialog box. This button is only enabled when a user-defined category is selected.

 Edit Selected Item

Displays the Edit Extended Property dialog box. This button is only enabled when a user-defined property is selected.

 Delete Selected Item

Displays the Delete Extended Property warning message box. This button is only enabled when a user-defined property is selected and that property is not referenced by another instance of the <SurveyFeatureClass>.

Property column

Lists the extended properties categories.

Description column

Displays the description of the extended property. This description can be assigned from the New/Edit Extended Property dialog box.

Depending on your *.sdx_def* file, each SurveyFeatureClass has relevant LandXML and User-defined collections. If you expand these collections, and select an attribute you can use the right-click short cut menu to edit or copy the property.

Survey Database

SurveyFeatureClass used to manage the extended properties of survey databases and projects.

Network

SurveyFeatureClass used to manage the extended properties of survey networks.

Network Group

SurveyFeatureClass used to manage the extended properties of Network Groups.

Figure

SurveyFeatureClass used to manage the extended properties of survey figures.

Figure Group

SurveyFeatureClass used to manage the extended properties of figure groups.

Survey Point

SurveyFeatureClass used to manage the extended properties of survey points.

Survey Point Group

SurveyFeatureClass used to manage the extended properties of survey point groups.

# New/Edit Extended Property Dialog Box

Use this dialog box to create a new property or edit an existing property in the Survey *.sdx* file.

XML Name

Specifies the XML name of the extended property.

Display Name

Specifies the display name of the property.

Description

Specifies a description of the property. This description is displayed in the Manage Extended Properties dialog box.

Property Field Type

Specifies the data type for the property. For more information on property field types, see Property Field Types Table (page 246).

Required Value

Select this check box to display a warning icon indicating that a required property has not been assigned to an instance of a <SurveyFeatureClass>.

Display In Survey User Interface

Select this check box to display the extended property in the Survey user interface. This check box is selected by default when you create a new user-defined property.

String Values

Specifies a list of string values if the Property Field type is set to String. You can enumerate the list of string values when you assign a value to a <SurveyFeatureClass>. Use the arrows to change the display order of the strings in this dialog box and in the Survey user-interface.

# New Network Group/Network Group Properties Dialog Box

Use this dialog box to create a new network group. To support LandXML, named collections of elements such as CgPoints or PlanFeature collections, Network Groups are added as a root collection within a Survey database item on the Survey tab in Toolspace.

## Network Group

Name

Specifies the name for the Network Group.

Description

Specifies a description for the Network Group.

## LandXML

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file that are relevant to networks are displayed. Click the cell in the Value column to assign an attribute.

## User-defined

If you are using the extended properties feature, and have created User-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

Add To Group

Select the check box to add the named network to the group

Name

Displays the name of the networks that can be added to the Network Group.

Description

Displays the description of the networks that can be added to the Network Group.

# New Figure Group/Figure Group Properties Dialog Box

Use this dialog box to create a new figure group. To support LandXML, named collections of elements such as CgPoints or PlanFeature collections, Figure Groups are added as a root collection within a Survey database item on the Survey tab in Toolspace.

### Figure Group

Name
    Specifies the name of the Figure Group.

Description
    Specifies the description of the Figure Group.

### LandXML

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

### User-defined

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

Add To Group
    Select the check box to add the figure to the new Figure Group.

Name
    Displays the name of the figure to add to the Figure Group.

Breakline
    Displays the status of the figure as a breakline.

Lot Line
    Displays the status of the figure as a lot line.

Layer
    Displays the layer the figure is drawn on.

Style
    Displays the figure style.

Site
    Displays the site to which the figure belongs.

Vertices
    Displays the number of vertices in the figure.

Length
    Displays the length of the figure.

Area
    Displays the area of the figure.

# New Survey Point Group/Survey Point Group Properties Dialog Box

Use this dialog box to create a new survey point group. To support LandXML, named collections of elements such as CgPoints or PlanFeature collections, Survey Point Groups are added as a root collection within a Survey database item on the Survey tab in Toolspace.

### Survey Point Group

Name
  Specifies the name of the Survey Point Group.

Description
  Specifies the description of the Survey Point Group.

### LandXML

If you are using the extended properties feature, the LandXML elements as defined in the *.sdx_def* file are displayed. Click the cell in the Value column to assign an attribute.

### User-defined

If you are using the extended properties feature, and have created user-defined properties they are displayed. Click the cell in the Value column to assign an attribute.

Add To Group
  Select the check box to add the survey point to the new Survey Point Group.

Number
  Displays the survey point number.

Name
  Displays the survey point name.

Easting
  Displays the easting of the survey point.

Northing
  Displays the northing of the survey point.

Elevation
  Displays the elevation of the survey point.

Description
  Displays a description of the survey point.

Longitude
  Displays the longitude of the survey point.

Latitude
  Displays the latitude of the survey point.

## Import / Re-import Point File Dialog Box

Import points into the survey database and survey network, as control points or non-control points, and specify options for linework processing.

Point File Format
  Displays the point file formats contained in the current drawing. The point file format determines the column order, delimiter, and coordinate zone for the file you are importing.

  > **NOTE** In linework processing, the point file format must have a Raw Description column.

Point File Name
  Displays the file path of the point file you want to import.

Point Type
  Specifies the imported points as either control points or non-control points in the network.

Current Figure Prefix Database
  Specifies the name of the prefix database. Select a database from the list.

Process Linework During Import
  Specify Yes to process linework during the file import.

Current Linework Code Set
  Specifies the linework code set that is used in the linework processing.

Process Linework Sequence
  Specifies the order in which survey point descriptions are processed to determine linework connectivity. Select one of the following:

  ■ By Import Order - processes points in the order in which they are imported. Point names are always processed by import order.

  ■ By Point Number - processes points sequentially, by point number (ascending order only).

Import Event Name
  Specifies the name of the import event that is created by the import command.

Import Event Description
  Optionally, specify a description for the import event.

Assign Offset To Point Identifiers
  Specifies that an offset is applied to point identifiers before the points are imported into the survey database.

Point Identifier Offset
  Specifies the offset value that is applied to the point identifier. For example, if the offset value is 1000, point number 101 is renumbered to 1101, or point name KC101 is renamed as KC101.1000.

Insert Network Object
  Specifies that the network object is inserted into the drawing when the import process is complete.

Insert Figure Objects
  Specifies that the figure object is inserted into the drawing when the import process is complete.

Insert Survey Points
  Specifies that the survey points are inserted into the drawing when the import process is complete.

# Import / Re-import Points From Drawing Dialog Box

Import points from the current drawing into the survey database and survey network, as control points or non-control points, and specify options for linework processing.

Point Type
  Specifies the imported points as either control points or non-control points in the network.

Current Figure Prefix Database
  Specifies the name of the figure prefix database. Select the database from the list.

Process Linework During Import
  Specify Yes to process linework during the file import.

Current Linework Code Set
  Specifies the linework code set that is used in the linework processing.

Process Linework Sequence

Specifies the order in which survey point descriptions are processed to determine linework connectivity. Select one of the following:

■ By Import Order - processes points in the order in which they are imported. Point names are always processed by import order.

■ By Point Number - processes points sequentially by point number (ascending order only).

Import Event Name

Specifies the name of the import event that is created by the import command. By default the import event name is the name of the imported file.

Import Event Description

Optionally, specify a description for the import event.

Assign Offset To Point Identifiers

Specifies that an offset is applied to point identifiers before the points are imported into the survey database.

Point Identifier Offset

Specifies the offset value that is applied to the point identifier. For example, if the offset value is 1000, point number 101 is renumbered to 1101, or point name KC101 is renamed as KC101.1000.

# Import Event Properties

Displays the properties of the selected event in the Import Events collection in the list view or the Import Events Panorama vista.

Name

By default, displays the name of the file that was imported.

Description

Displays an optional description.

Date/Time

Read-only property that is assigned at the time of the import and updated when there is a re-import.

User Name

DIsplays the read-only property of the user's Windows login name that is used at the time of the import and updated when there is a re-import.

File

Displays the assigned name of the imported file. (Read-only).

Import Type

Displays one of the read-only following import types:

■ Field Book File

■ Survey LandXML

■ Point File

■ Points From Drawing

Point Identifier Offset

Displays the offset value that was specified in the import command.

Point File Format

Displays the type of point file format used during the Import Point File command, or specified during the import survey data command.

Linework Code Set

Displays the name of the linework code set.

Process Linework Sequence

Displays the order in which survey point descriptions are processed to determine linework connectivity.

- By Import Order - processes points in the order in which they are imported. Point names are always processed by import order.

- By Point Number - processes points sequentially by point number (ascending order only).

Equipment Database

Displays the name of the equipment database associated with the import event.

Equipment

Displays the name of the equipment associated with the import event.

Figure Prefix Database

Displays the name of the figure prefix database associated with the import event.

# Import Events Editor

Use the Import Events Editor vista to access all events associated with a survey network.

The Import Events Editor displays the properties for each event. Right-click an individual event to access the shortcut menu commands.

# Process Linework Dialog Box

The Process Linework command processes linework connectivity between points, using a specified linework code set in situations where a linework code set was not previously specified.

You can also reprocesses linework connectivity after you make corrections to the survey point descriptions in the Survey Point Properties dialog box (page 2416).

Current Figure Prefix Database

Specify the figure prefix database.

Current Linework Code Set

Specify the linework code set.

Process Linework Sequence

Specifies the order in which survey point descriptions are processed to determine linework connectivity. Select one of the following:

- By Import Order - processes points in the order in which they are imported. Point names are always processed by import order.

- By Point Number - processes points sequentially by point number (ascending order only).

Insert Figure Objects

Inserts figures in the current drawing.

Insert Survey Points

Inserts survey points in the current drawing.

**Related procedures:**

■ Processing Survey Linework (page 251)

# New Linework Code Set Dialog Box

Specify the name for the linework code set.

Enter a unique name for the linework code set.

# Edit Linework Code Set Dialog Box

Specify the syntax of the field code for a survey point description.

This syntax is interpreted when you process linework by importing or re-importing survey data, or use the Process Linework command.

For more information, see Examples - Linework Codes (page 231).

### Information

Name

Specifies the name of the linework code. To re-name the linework code, enter a new name and click OK or Apply.

Description

Specifies a description for the linework code.

### Coding Methods

**Codes**: All codes in the linework code must be unique. If you create a duplicate code, a warning icon is displayed on both the category and the property. Before you can close the Edit Linework Code Set dialog box, you must resolve all duplicate codes.

**Undefined codes**: If a code has no assigned value (empty string/blank value) it is considered unused and not defined. Undefined codes may be necessary in situations where you are using an older model data collector that may not have enough physical keys to support the codes.

---

**WARNING** It is recommended that you use a <space> as the Feature /Code Delimiter Property Value. Using decimal characters in the Feature/Code Delimiter Property Value may cause certain codes to fail. For example the . (decimal) character is valid for a right-turn value and the - (minus sign) character is valid for a negative right-turn value.

---

For more details on linework code properties, see Examples - Linework Codes (page 231).

Feature/Code Delimiter

Specifies the code that separates the feature name from the code. It is recommended that you use a space (" ") character.

Field Code Escape

Specifies the code that stops the evaluation of the field code. Characters displayed after the escape character are considered comments. It is recommended that you use a / (slash) character.

### Special Codes

Begin

Specifies the code that indicates a new figure starts at the specified point.

Continue

Specifies the code that indicates a figure continues from its last vertex.

End

Specifies the code that terminates the specified figure.

Close

Specifies the code that continues an active figure from its last vertex to the current point and closes the figure with a line segment back to the starting vertex for the figure.

Horizontal Offset

Specifies the code that indicates a relative horizontal offset for creating a parallel figure starting at the current point, or transitioning to another offset if the active figure has a previous horizontal offset. A value must follow the Horizontal offset code. A negative value offsets the active figure to the left and a positive value offsets the active figure to the right.

Vertical Offset

Specifies the code that indicates a relative vertical offset starting at the current point. A value must follow the Vertical offset code. A positive value is added to the elevation of the current vertex for the active figure and a negative value is subtracted from the current vertex for the active figure.

Stop Offset

Specifies the code that terminates all offsets on the active figure, and allows only the active figure to be continued.

## Line Segment Codes

Recall Point

Specifies the code that connects the active figure with a segment from the last point, or a specified pointID.

Connect Point

Specifies the code that creates a new figure (of the same feature) with a single line segment from the current point to the specified point ID. The specified point ID follows the Connect point code.

Rectangle

Specifies the code to offset the segment coming into the current point by the specified number.

Right Turn

Specifies the code for a <Right turn> line segment that allows for additional vertices to be inserted into the figure, perpendicular or extensions (straight offsets). Offset values, <Extend>, or <Rectangle> codes follow the <Right turn> code. Positive numbers indicate a turn to the right and negative numbers indicate a turn to the left.

Extend

Specifies the code that with a number is used to make an extension of a line segment ahead through the current point by using a positive value or short of the current point by using a negative value, for example <Figure Name><Delimiter><Extend><Value>.

**NOTE** The Extend code can be used within the Right turn code.

## Curve Segment Codes

Begin Curve

Specifies the code that indicates the current point is the beginning of a curve (arc) segment. The current point with this code is the first point on the curve. The next point with the same figure name is considered a point on the curve, and third point with the same figure name is the curve endpoint.

### End Curve

Specifies the code that is used with the <Begin curve> code and is used to define curved segments with more than three points. Starting at the point with the <Begin curve> code, the linework processor will look for a <End curve> code. If the <End curve> code is found, all the points between the <Begin curve> code and <End curve> code are used to fit arc segments of the figure, such that the curves pass through each of the points (similar to the PEDIT/Fit [curve] option).

### Circle

Specifies the code that stops the linework on the previous point (an implied <End> code) and causes the current point to create a circle in one of three methods.

### Point on Curve

Specifies the code to indicate that the current point resides on a curve segment.

# Import Survey Data Wizard

Use this wizard to import field book files, point files, Survey LandXML files, or selected points in a drawing. At the completion of the import an import event is created.

For more information, see Import Events (page 249).

## Import Survey Data Wizard - Specify Database Page

Specify an existing survey database or create a new one.

### Survey Databases

Select an existing survey database.

### Create New Survey Database

Displays the New Local Survey Database dialog box where you can enter the folder name.

### Edit Survey Database Settings

Displays the Survey Database Settings dialog box.

## Import Survey Data Wizard - Specify Data Source Page

Specify the data source type and select the data.

### Data Source Type

- Field Book File

- LandXML file

- Point File

- Points From Drawing

### Source File

Browse to the source file and select the file you want to import.

## Import Survey Data Wizard - Specify Network Page

Select an existing survey network or create a new network.

### Survey Network

Displays the existing networks in the survey database.

Create New Network

Displays the New Network dialog box where you can enter a name and description for the network.

## Import Survey Data Wizard - Import Options Page

Specify the import settings for the selected data source.

■ Field Book File: Import / Re-import Field Book Dialog Box (page 2406)

■ LandXML File: Import / Re-import Survey LandXML Data Dialog Box (page 2420)

■ Point File: Import / Re-import Point File Dialog Box (page 2429)

■ Points From Drawing: Import / Re-import Points From Drawing Dialog Box (page 2430)

# Tables Dialog Boxes

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Use the following links to access information about the Table dialog boxes.

## Table Style Dialog Box

Use this dialog box to create or edit table styles.

## Information Tab (Table Style Dialog Box)

Use this tab to change the table style name and description information.

Name
Specifies a name for the style.

Description
Specifies a description for the style.

Created By
Displays the AutoCAD login name of the person who created the style.

Date Created
Displays the date and time the table style was created.

Last Modified By
Displays the AutoCAD login name of the person who last modified the style.

Date Modified
Displays the date and time the table style was last modified.

**Related procedures:**

■ Defining a New Table Style (page 1556)

## Data Properties Tab (Table Style Dialog Box)

Use this tab to format table text style and structure.

## Table Settings

Wrap Text

Forces text to fit within a specified column width by wrapping it in multiple lines.

**NOTE** Wrap text applies to columns that are not set to automatic width.

Maintain View Orientation

Specifies whether the table is realigned if the drawing view is rotated or twisted.

- **Selected**: Realigns the table to the view direction.

- **Cleared**: Does not realign the table.

Repeat Title In Split Tables

Specifies whether titles are duplicated in split tables.

- **Selected**: Duplicates the title in all table sections.

- **Cleared**: Creates split tables without titles on the second and additional sections.

Repeat Column Headers In Split Tables

Specifies whether column headers are duplicated in split tables.

- **Selected**: Duplicates the column headers in all table sections.

- **Cleared**: Creates split tables without headers on the second and additional sections.

Sort Data

Specifies whether table data is sorted based on a selected column.

- **Selected**: Sorts table data based on the specified column.

- **Cleared**: Orders table contents in the order in which the objects are drawn.

Sorting Column

Specifies the column to sort.

Order

Specifies how to sort the column.

- **Ascending**: Sorts from lowest to highest.

- **Descending**: Sorts from highest to lowest.

## Text Settings

Title Style

Specifies a text style for the title.

Header Style

Specifies a text style for the column headings.

Data Style

Specifies a text style for the row data.

Height

Specifies the height values for Title, Header, and Data text.

## Structure

Controls the composition of data. You can do the following:

■ Edit the title or column headings. Double-click the text to display the Text Component Editor (page 1964).

■ Edit the column widths (see Column Width below).

■ Edit the content of the data columns. Double-click a Column Value cell to display the Text Component Editor (page 1962).

■ Rearrange the order of columns. Select the column and drag it. The first column is reserved as the tag column and cannot be moved.

## Table Headers

Table Title (for example, "Line Table")

Specifies the title contents. To edit the title, double-click the text (such as "Line Table") to display the Text Component Editor. For more information, see Text Component Editor (page 1964).

Column Title

Specifies the column heading contents. To edit the headings, double-click the text to display the Text Component Editor.

**NOTE** Except for alignment table styles, the Properties tab in the Text Component Editor is disabled when you edit column heading text.

## Columns

Column Width

Specifies column widths. Click ![icon] and select one of the following options:

■ ![icon] **Automatic**: Sizes the column width according to the longest text string in the table.

■ ![icon] **Manual**: Sets the column width according to a specified character width.

**NOTE** When you specify a fixed width and do not select the Wrap Text option, and the content exceeds the column width, the value in the column is a series of #### characters equal to the width of the column.

Column Value

Specifies the content for each table column. The content can be derived from a combination of static text and property fields associated with a feature.
To edit the column values, double-click a cell in the row. Use the Properties (page 1962) tab in the Text Component Editor to define the column's dynamic content.

![plus icon]

Adds a column at the end of the table. Because the new column is empty, you must define a column heading, width, and value. The column width is set to Automatic width by default.

![delete icon]

Deletes a column. Select the column you want to delete and then click the icon.
The first column cannot be deleted.

**Related procedures:**

■ Table Text Properties (page 1556)

-

-

-

-

# Display Tab (Table Style Dialog Box)

Use this tab to determine values for components in a table.

## View Direction

Specifies the view direction for defining display settings.

2D

Selects the 2D set of display properties for viewing and editing.

3D

Selects the 3D set of display properties for viewing and editing.

## Component Display

Component Type

Specifies display settings for each table component.

- **Overall Border**: Graphic line that forms a boundary around the outside of table.

- **Title Separator**: Horizontal graphic line that separates title from column headings.

- **Header Separator**: Horizontal graphic line that separates column headings from data rows.

- **Data Separator**: Horizontal graphic line that separates data rows.

- **Data Divider**: Vertical graphic line that separates columns.

- **Title Area Fill**: Solid hatch pattern that serves as background fill for the table title area.

- **Header Area Fill**: Solid hatch pattern that serves as background fill for the column heading areas.

- **Data Area Fill**: Solid hatch pattern that serves as background fill for all data rows.

- **Title Text**: Table name in first row.

- **Header Text**: Column names in second row.

- **Data Text**: All text in data rows.

Visible

Specifies whether the component is visible. Click 💡 to switch between visible and hidden 💡 .

Layer

Specifies the layer from which the component derives its layer properties. Click the column to display the Layer Selection dialog box.
If you set the layer to zero (0), the components use the layer properties as specified in the Drawing Settings dialog box.

Color

Specifies the color of the component. Click the column to display the Select Color dialog box.

Linetype
   Specifies the linetype of the component. Click the column to display the Select Linetype dialog box.

LT Scale
   Specifies the linetype scale of the component. To change the linetype scale for an object component, highlight the linetype scale and enter a new value in the cell.

Lineweight
   Specifies the lineweight of the component. Click the column to display the Lineweight dialog box.

Plot Style
   Specifies a set of overrides to be used when plotting.

**Related procedures:**

■ Table Display (page 1563)

## Summary Tab (Table Style Dialog Box)

Use this tab to review style settings. The property window contains two columns: Property and Value.

You can make changes to values in the Value column of the Summary tab; however, the display settings are not available on this tab.

**Related procedures:**

■ Viewing a Summary of Table Settings (page 1563)

# Table Creation Dialog Box

Use this dialog box to add tables to a drawing.

Table Style
   Specifies the table style to use.

 Style Selection Menu
   Creates a new table style, edits a table style, or selects a table style from the drawing.

 Style Detail
   Displays the Style Detail dialog box.

Table Layer
   Specifies the layer on which the table is drawn. Click  to select a layer.

**Selection**

Controls what data is inserted into the table.

---

NOTE Because objects must be labeled to create a table that contains their data, most tables require you to specify the table data by selecting a label style or a label in the drawing. Exceptions are point tables and alignment segment tables, which allow you to specify point groups and alignments.

---

Label Style Name
   Adds data to the table from all objects labeled with the selected label style. Select the check box next to the style name to select a label style.

Selection Rule

Controls whether existing or new data is added to the table. After selecting the Apply check box, click the Selection Rule field and select one of the following options.

- **Add Existing**: Creates a table from existing objects labeled with the style. Data from new objects labeled with the style is not added to the table.

- **Add Existing and New**: Creates a table from existing and new objects that are labeled with the style.

By Alignment (alignment segment tables only)

Adds data from the selected alignment to the table.

⌖ Select Point Groups (point tables only)

Adds data from the selected point group to the table.

▣ Pick On-Screen

Adds data to the table from the objects or labels that you select in the drawing.

**NOTE** This area of the Table Creation dialog box is not displayed for the Tangent-Tangent Intersection (PI) table.

## Split Table

Split Table Option

- **Selected**: Divides a table into two or more sections after a specified maximum number of rows has been met.

- **Cleared**: Does not divide the table.

Maximum Rows Per Table

Specifies the maximum number of rows to include per section.
If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of sections to include in each stack. See also the following Tile Tables option.

Offset

Specifies the gap between split tables.

Tile Tables

Specifies the direction in which tables are stacked.

- **Across**: Places split tables in a row from left to right. When the maximum tables per stack is reached, a new row is started.

- **Down**: Places split tables in a column from top to bottom. When the maximum tables per stack is reached, a new column is started.

## Behavior

Reactivity Mode

Specifies whether tables are updated automatically.

- **Static**: Keeps table data fixed. If you want to see changes, click Update Table from the shortcut menu.

- **Dynamic**: Updates table data dynamically when you make changes.

# Table Properties Dialog Box

Use this dialog box to review table properties and to change styles and appearance.

Table Style

Specifies the table style to use.

 Style Selection Menu

Creates a new table style, edits a table style, or selects a table style from the drawing.

 Style Detail

Displays the Style Detail dialog box.

## Split Table

Split Table Option

■ **Selected**: Divides a table into two or more sections after a specified maximum number of rows has been met.

■ **Cleared**: Does not divide the table.

Maximum Rows Per Table

Specifies the maximum number of rows to include per section.
If the number of data rows exceeds the specified maximum, the table is split into sections, and they are displayed either side by side (left to right), or stacked vertically.

Maximum Tables Per Stack

Specifies the maximum number of sections to include in each stack. See also the following Tile Tables option.

Offset

Specifies the gap between split tables.

Tile Tables

Specifies the direction in which tables are stacked.

■ **Across**: Places split tables in a row from left to right. When the maximum tables per stack is reached, a new row is started.

■ **Down**: Places split tables in a column from top to bottom. When the maximum tables per stack is reached, a new column is started.

 Force Realignment Of Stacks

Realigns the stacks in a split table.

## Behavior

Reactivity Mode

Specifies whether tables are updated automatically.

■ **Static**: Keeps table data fixed. If you want to see changes, click Update Table from the shortcut menu.

■ **Dynamic**: Updates table data dynamically when you make changes.

 **Force Content Update**

Refreshes the table if the Reactivity Mode is set to Static.

**Related procedures:**

■ Changing Table Properties (page 1567)

# Renumbering Dialog Box

Use this dialog box to set the starting number and increment value for renumbering.

This dialog box is used for renumbering parcels and for renumbering alignment tags.

Site Name

Specifies the site in which the parcels, parcel tags, or alignment tags are renumbered. Click  to select a site in the drawing by selecting an object in the site.

## Increment Format

Starting Number

Specifies the next number that is used for renumbering the objects or tags.

Increment Value

Specifies the difference between numbers in the numbering sequence.

**Related procedures:**

■ Renumbering Label Tags (page 1544)

# Add Selection Dialog Box

Adds content to a table.

Use the Add Selection dialog box to add content to a table.

Append to Current Selection Set

Controls what data is added to the table.

---

**NOTE** Because objects must be labeled to create a table that contains their data, most tables require you to specify the table data by selecting a label style or a label in the drawing. Exceptions are point tables and alignment segment tables, which allow you to specify point groups and alignments.

---

Label Style Name

Adds data from all objects labeled with the selected label style to the table. Select the Apply check box to select a label style.

Selection Rule

Controls whether existing or new data is added to the table. After selecting the Apply check box, click the Selection Rule field and select one of the following options.

■ **Add Existing**: Creates a table from existing objects labeled with the selected style. Any new objects labeled with the specified style are not added to the table.

■ **Add Existing and New**: Creates a table from existing and new objects that are labeled with the selected style.

Pick On-Screen

Adds data selected in the drawing to the table.

 Select Point Groups (point tables only)

Adds data from the selected point group to the table.

**Related procedures:**

■  Changing Table Data (page 1568)

# Remove Selection Dialog Box

Deletes rows from a table.

Use the Remove Selection dialog box to remove rows from a table.

Exclude From Current Selection Set

Controls what data is removed from the table.

Label Style Name

Removes data for all objects labeled with the selected label style. Select the Apply check box to select a label style.

Exclusion Rule

**Remove All**: Removes all data currently labeled with the specified label style.

 Pick On-Screen

Removes selected table rows.

 Select Point Groups (point tables only)

Removes data from the selected point group from the table.

**Related procedures:**

■  Changing Table Data (page 1568)

# Replace Selection Dialog Box

Replaces table data.

Use the Replace Selection dialog box to replace table content with other data.

Replace Current Selection Set

Controls what data is added to the table, replacing all the data that is currently in the table.

Label Style Name

Adds data to the table from all objects labeled with the selected label style. Select the Apply check box to select a label style.

Exclusion Rule

Controls whether existing or new data is added to the table. After selecting the Apply check box, click the Selection Rule field and select one of the following options.

- **Add Existing**: Creates a table from existing objects labeled with the selected style. Any new objects labeled with the specified style are not added to the table.

- **Add Existing and New**: Creates a table from existing and new objects that are labeled with the selected style

Pick On-Screen

Adds data selected in the drawing to the table.

Select Point Groups (point tables only)

Adds data from the selected point group to the table.

**Related procedures:**

- Changing Table Data (page 1568)

# Utilities Dialog Boxes

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Use the following links to access information about the Utilities dialog boxes.

## Description Dialog Box

Use this dialog box to change the description of an object.

Description Box
Specifies the description of the object.

## Mapcheck Analysis Window

Use this window to create, edit, and output a Mapcheck Report.

Collapse All
Collapses all items in the tree view.

Expand All
Expands all items in the tree view.

Use Command Line Interface
Toggles on and off the command line input. When this button is toggled on you are prompted to pick labels in the drawing to create the mapcheck. Toggle this button off to create a mapcheck by entering data manually.

New Mapcheck Report
Clears the existing mapcheck report and starts a new report.

New Mapcheck
Creates a new mapcheck and Point of Beginning (POB) in the Mapcheck Report.

New Side
Adds a new side to the end of the current mapcheck.

---

**NOTE** This button is disabled if you have not entered a POB.

---

 Insert Side Before

Inserts a new side before the current side.

 Insert Side After

Inserts a new side after the current side.

---

**NOTE** This button is disabled if a side is not selected, or if you have not entered a POB.

---

 Delete

Deletes the current side or current mapcheck if no side is current.

 Input View

Displays the Mapcheck Analysis window in input mode. You can enter and modify values in the view.

 Output View

Displays the Mapcheck Analysis window in output mode. This view displays the closure summary.

 Copy to Clipboard

Copies the contents of the input view or output view to the Windows Clipboard.
If a mapcheck is current then only that mapcheck is copied to the clipboard. If no mapcheck is current then all mapchecks are copied. Press ESC to clear a selection in tree view.

 Insert MTEXT

Inserts the contents of the input view or output view as an Mtext object in the drawing.
If a mapcheck is current then only that mapcheck is inserted into the drawing. If no mapcheck is current then all mapchecks are inserted into the drawing as separate Mtext objects. Press ESC to clear a selection in tree view.

 Create Polyline

Inserts a polyline into the drawing representing the current mapcheck. If there is no current mapcheck then a polyline for all mapchecks is inserted into the drawing. Press ESC to clear a selection in tree view.

**Mapcheck Tree**

Name

Specifies the optional mapcheck name. If a name is not assigned, the display is Mapcheck <number>.

 Point of Beginning (POB)

Click to specify the Point Of Beginning (POB) in the drawing. All other Side points are calculated from the POB.

Easting

Modify the easting value for the POB.

Northing

Modify the northing value for the POB.

**Side**

A side can be either a line or a curve. The following section describes the values for a line Side type.

Side Type

Specifies the Side as line.

Angle Type

Specifies the angle type to be used. Select Direction, Angle, or Deflection from the drop-down list.

**Angle**

Specifies the angle value for the Side based on the angle type.

**Reverse Direction**

Select the check box to reverse the line direction or the curve chord direction by 180 degrees to calculate the endpoint coordinates for the Side.

**Distance**

Specifies the distance value for the Side.

**Easting**

Displays the calculated easting value for the Side.

**Northing**

Displays the calculated northing value for the Side.

A side can be either a line or a curve. The following section describes the values for a curve Side type.

**Side Type**

Specifies the Side as a curve.

**Curve Direction**

Specifies the curve directions as Clockwise or Counter-clockwise.

**Traverse Method**

Specifies how the error of closure is calculated and displayed in the Output View.
Specify one of the following:

- Across Chord: The endpoint of the curve is calculated from the Chord direction and distance.

- Through Radius: The endpoint of the curve is calculated based on the direction to the radius point, the radius length, the delta angle, and the direction from the radius point to the point of tangency. If this option is selected, additional information is displayed in the Output View.

**Radius**

Specifies the radius value. This value is required.

---

**NOTE** Calculated curve properties do not display in the Input View. Click  Output View to display the output view.

---

**Arc Length**

Specifies the length of the curve.

**Delta Angle**

Specifies the curve delta angle.

**Tangent**

Specifies the curve tangent distance.

**Chord Angle Type**

Specifies the angle type as Direction, Angle, or Deflection. If the angle type is set to Angle or Deflection, the specified angle value is relative from a tangent line at the point of curvature to the point of tangency (long chord).

**Chord Angle**

Specifies the chord angle value.

Reverse Direction

Select the check box to reverse the line direction or the curve chord direction by 180 degrees to calculate the endpoint coordinates for the Side.

Chord Distance

Specifies the chord distance value.

Easting

Displays the calculated easting value of the Side endpoint.

Northing

Displays the calculated northing value of the Side endpoint.

**Related procedures:**

■ Performing a Mapcheck Analysis (page 1608)

# Adjust Mapcheck Analysis Dialog Box

Use this dialog box to adjust a mapcheck report for closure.

Do Angle Balance

Select Yes to perform a mathematical analysis based on the method you specify as the Horizontal Adjustment Method.

Horizontal Adjustment Method

Specify one of the following traverse analysis methods:

■ Compass Rule: A method of corrections where the closing errors are assumed to be as much due to errors in observed angles as by the errors in measured distances. The closing errors in latitude and departure are distributed according to the ratio of the length of the line to the total length of the traverse

■ Transit Rule: A method of balancing a traverse where the closing errors are assumed to be caused less by the errors in the observed angles than by the errors in measured distance. Corrections are distributed according to the ratio of the latitude and departure of each leg of the traverse to the sums of the latitude and departures of the entire traverse.

■ Crandall Rule: A method of balancing a traverse where all the angular error is distributed throughout the traverse and all adjustments to the traverse are due to modifying the traverse distances. The modification distance made to each leg is such that the sum of the squares is a minimum. Corrections corresponding to the closing errors assume that the closing errors are random and normally distributed, and that all the angular error has been adjusted prior to the adjustment routine.

■ Least Squares: A method of balancing a traverse. The squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method weights the individual measurements according to the specifications set in the Equipment correction settings to determine the source of error. You can adjust the data for an individual traverse loop, or a traverse network, if located from multiple observations.

**NOTE** The Least Squares option uses the specified values for the Standard Deviations in the Equipment Properties for the current Survey Equipment Database. Make sure that values are appropriate for the least squares method of adjustment. For more information, see New Equipment/Equipment Properties Dialog Box (page 2384).

Horizontal Closure Limit

Specifies the minimum allowable error of closure in the horizontal direction, for example a value of 15000.00 is 1 part in 15000.

Angle Error Per Set
    Specifies the maximum angle error.

Update Mapcheck
    Specify Yes to update the mapcheck report with the adjusted coordinates.

---

**NOTE** In the Mapcheck Analysis Input View, the 🔧 icon is displayed next to the Mapcheck name indicating the mapcheck has been adjusted. To clear an adjustment, right-click the Mapcheck ➤ Clear Adjustment.

---

**Related procedures:**

■ Adjusting a Mapcheck Analysis (page 1614)

# Notes Dialog Box

Use this dialog box to attach notes or reference documents to an object.

# Notes Tab (Notes Dialog Box)

Use this tab to attach notes to an object.

Notes Text Box
    Specifies comments to associate with the object.

**Related procedures:**

■ Attaching Text to an Object (page 1616)

# Reference Docs Tab (Notes Dialog Box)

Use this tab to attach reference documents to an object.

Document Column
    Specifies the reference document name.

Type Column
    Displays the file type for the attached reference file.

Description Column
    Displays the file description for the attached reference files.

Add
    Displays the Select Reference Document dialog box. Browse to select the reference file that you want to attach.

Edit
    Displays the Reference Document dialog box. Click File to browse for another document. You can edit the reference document description in the Description box.

Delete
    Deletes the selected reference document.

**Related procedures:**

■ Attaching External Reference Documents to an Object (page 1616)

# Object Viewer Dialog Box

Use this dialog box to control the view of selected objects in your drawing.

The Object Viewer contains the ViewCube and the SteeringWheel. For more information, see Use ViewCubeand Navigation Wheelsin the AutoCAD Help.

### ![icon] Save Image

Saves the current view as a separate file. The supported file formats are PNG, JPG, BMP, TIF.

### Visual Style List

Specifies the object appearance style. Set the style to one of the following:

- 3D Hidden: Displays objects in the current view in a 3D wireframe representation with all visible lines displayed as continuous and all hidden lines displayed as dashed.

- 3D Wireframe: Displays objects in the current view in a 3D wireframe representation with all lines displayed, including those hidden by other objects.

- Conceptual: Displays objects in a transparently shaded 3D view with all lines visible.

- Realistic: Displays objects in a shaded 3D view.

### ![icon] Pan

Moves the current view without changing its size.

### ![icon] Zoom

Increases or decreases the apparent size of objects in the current viewport.

### ![icon] SteeringWheel

Provides a Navigation Wheels menu that is divided into different sections known as wedges. Each wedge on the wheel represents a single navigation tool. You can pan, zoom, or manipulate the current view of a model in different ways.

### ![icon] 3D Orbit

Sets the display window to 3D orbit view.

**NOTE** Hold down Shift and click to pan in the Object Viewer. Hold down Ctrl and click to zoom in the Object Viewer.

### ![icon] Adjust Distance

Displaces objects a specified distance in a specified direction. This control is available only when the Perspective control is selected.

### View Control List

Specifies the current view. Set the view to either Top, Bottom, Left, Right, Front, or Back. Set the isometric view to either SW Isometric, SE Isometric, NE Isometric, or NW Isometric

### ![icon] Parallel

Sets the view to a parallel plane.

### ![icon] Perspective

Sets a perspective view and makes the Adjust Distance and Lens Length controls available.

 Lens Length

Sets the length of the lens used in perspective viewing for the current viewport. This control is available only when the Perspective control is selected.

 Zoom Window

Zooms to display an area specified by a rectangular window.

 Zoom Center

Zooms to display a window defined by a center point.

 Zoom Extents

Zooms to display the drawing extents and results in the largest possible display of all the objects in the window area.

 Zoom In

Zooms in an increment as defined by the Zoom Factor.

 Zoom Out

Zooms out an increment as defined by the Zoom Factor.

 Zoom Factor

Sets the zoom increments used by the Zoom In and Zoom Out commands.

 Set View

Sets the view in the drawing equal to the view in the Object Viewer.

Display Configuration List

Specifies the object viewer display configuration.

NOTE AutoCAD Civil supports only the Standard Display Configuration.

**Related procedures:**

■   Using the Object Viewer (page 1617)

# AEC Editor Tab (Options Dialog Box)

Use the AEC Editor tab in the Options dialog box to control the visibility of the design criteria warning marker, and settings that affect the Export To AutoCAD commands.

**Solution Tip**

Controls the visibility of the design criteria warning marker. This warning marker is displayed in the drawing window when an alignment or profile entity violates the specified design criteria.

NOTE If either of these check boxes are cleared, the warning marker will not be displayed on any alignment or profile objects that violate the specified design criteria. However, the alignments and profiles will still be validated against the specified design criteria, and warning markers will appear the Alignment Entities and Profile Entities vistas and the Alignment Layout Parameters and Profile Layout Parameters dialog boxes.

Drafting

Displays the warning marker in the drawing window.

Plotting/Publishing

Displays the warning marker in plotted or published output.

## Export to AutoCAD

Bind Xrefs

Binds xrefs to the host drawing when using the Export To AutoCAD commands, creating local block definitions. If you select this option, all xrefs and their dependent named objects become a part of the exported drawing.

Use Insert Method When Binding Xrefs

- **Selected**: The drawing names of the xref drawings are discarded when the exported drawing is created. In addition, the xref layers and styles are incorporated into the host drawing, and layers with the same name acquire the properties of the layers of the host drawing.
  This option binds the xref to the current drawing in a way similar to detaching and inserting the reference drawing. Rather than being renamed using *blockname$n$definitionname* syntax, xref-dependent named objects are stripped of the xref name. As with inserting drawings, no name-incrementing occurs if a local named object shares the same name as a bound xref-dependent named object. The bound xref-dependent named object assumes the properties of the locally defined named object. For example, if you have an xref named FLOOR1 containing a layer named WALL, after binding with the Insert option, the xref-dependent layer FLOOR1|WALL becomes the locally defined layer WALL.

- **Cleared**: The xref identities, such as layer names, are retained when you export a file to AutoCAD. By clearing this check box, xref-dependent named objects are changed from *blockname|definitionname* to *blockname$n$definitionname* syntax. In this manner, unique named objects are created for all xref-dependent definition tables bound to the current drawing.

  For example, if you have an xref named FLOOR1 containing a layer named WALL, after binding the xref, the xref-dependent layer FLOOR1|WALL becomes a locally defined layer named FLOOR1$0$WALL. The number $n$ is automatically incremented if a local named object with the same name already exists. In this example, if FLOOR1$0$WALL already existed in the drawing, the xref-dependent layer FLOOR1|WALL would be renamed FLOOR1$1$WALL.

Filename Prefix

Adds the specified prefix to a file created with the Export To AutoCAD commands.

Filename Suffix

Adds the specified suffix to a file created with the Export To AutoCAD commands.

## Block Properties of Exploded Object

Maintain Resolved Layer, Color, Linetype

Maintains the layer, color, and linetype of AutoCAD Civil objects when you explode them.
When you explode an object, you create several primitive objects grouped in a block definition. If this option is selected any objects whose component layer, color, and linetype properties are set to ByBlock take the layer, color, and linetype of the parent object. If this option is cleared, when you explode an object, properties that are ByBlock remain ByBlock. Clear this option if you want to explode the block definition even further.

## Unselected Grip Colors

Controls the colors of several types of unselected grips.

Object

Specifies the color of the AutoCAD Civil unselected object grips. For example, if you click on a profile, the unselected grips use this color.

Secondary Object

Not used by AutoCAD Civil.

Style

Not used by AutoCAD Civil

Auxillary

Not used by AutoCAD Civil

---

**NOTE** The other settings on this tab are not used by AutoCAD Civil.

**Related procedures:**

■ Changing the AEC Editor Settings (page 1658)

■ Viewing and Correcting Alignment Design Criteria Violations (page 1008)

■ Viewing and Correcting Profile Design Criteria Violations (page 1103)

# Event Viewer Vista

Use the Event Viewer to view messages that are logged during an AutoCAD Civil session.

## Menu and Toolbar

Action menu

Contains general Event Viewer commands, including commands for working with log files, exporting an event list, and listing event properties. Click a command on the menu.

View menu

Contains commands that control what events are displayed in the event list and how they are displayed. Click a command on the menu.

⇐

Returns to a previously selected collection in the event tree.

⇒

Moves forward to a previously selected collection in the event tree.

▢

Makes the Event Viewer collection in the event tree active, displaying all events in the event list.

▤

Shows or hides the event tree.

⚠

Displays the Event Properties (page 2456) dialog box for the selected event.

↻

Refreshes the Event Viewer.

▤→

Exports the event list to a file. For more information, see Exporting an Event List to a Text File (page 1630).

**Event Tree**

Tree

Controls the events that are listed in the event list. Expand the tree and select a collection to list the events in the collection. For more information, see The Event Viewer Vista (page 1626).

**Event List**

Type

Specifies the type of event: either Information, Warning, or Error.

Date

Specifies the date the event was logged to the Event Viewer.

Time

Specifies the time the event was logged to the Event Viewer.

Source

Specifies the feature that logged the event to the Event Viewer.

User

Specifies the user who performed the action that caused the event to be logged to the Event Viewer.

Description

Describes the event.

> **NOTE** You can control the columns and the events that are displayed in the event list. For more information, see Controlling the Columns Displayed in the Event Viewer (page 1627) and Controlling the Events Displayed in the Event Viewer (page 1628).

**Related procedures:**

■ The Event Viewer Vista (page 1626)

# Event Properties Dialog Box

Use this dialog box to display detailed information about a specific event in the Event Viewer and to copy the information to the Windows clipboard.

Type

Specifies the type of event: either Information, Warning, or Error.

Date

Specifies the date the event was logged to the Event Viewer.

Time

Specifies the time the event was logged to the Event Viewer.

Source

Specifies the feature that logged the event to the Event Viewer.

User

Specifies the user who performed the action that caused the event to be logged to the Event Viewer.

Description

Describes the event.

⬆ Up Arrow

Changes the event displayed in the dialog box to the previous event in the event list.

⬇ Down Arrow

Changes the event displayed in the dialog box to the next event in the event list.

 Copy to Clipboard

Copies the contents of the Event Properties dialog box so that it can be pasted into another application.

**Related procedures:**

■ The Event Viewer Vista (page 1626)

# Filter Events Dialog Box

Use this dialog box to control the events that are listed in Event Viewer event list.

Event Types

Specifies the types of events to be included in the event list. Select the check box next to any or all of the following event types:

■ Information

■ Warning

■ Error

Event Source

Displays only the events logged by the specified source. Select the name of a source in the list or click All to display all events, regardless of their source.

User

Displays only the events associated with the specified user. Select the name of a user in the list or click All.

**NOTE** You can see more than one user name in the event list if you open a log file that contained events that were created by another user and you then perform actions that cause events to be logged using your login name.

**From Event**

From list

Specifies the date and time of the first event displayed in the event list. In the list, select one of the following:

■ First Event: Begins the display with the first event in the event list.

■ Events On: Begins the display with the first event in the list that falls on or after the specified date and time.

From Date

When Events On is specified, specifies the date for the first event in the event list. Do one of the following:

■ Enter a date.

■ Click the list arrow to display a calendar. Click Today at the bottom of the calendar to set the date to the current date. Click the right and left arrows to specify a month. Click a day on the calendar to specify a day.

From Time
When Events On is specified, specifies the time for the first event in the event list.

**To Event**

To list
Specifies the date and time of the last event displayed in the event list. In the list, select one of the following:

■ Last Event: Ends the display with the last event in the event list.

■ Events On: Ends the display with the last event in the event list that falls on or before the specified date and time.

To Date
When Events On is specified, specifies the date for the last event in the event list. Do one of the following:

■ Enter a date.

■ Click the list arrow to display a calendar. Click Today at the bottom of the calendar to set the date to the current date. Click the right and left arrows to specify a month. Click a day on the calendar to specify a day.

To Time
When Events On is specified, specifies the time for the last event in the event list.

**Related procedures:**

■ Customizing the Event Viewer (page 1627)

# Find Event Dialog Box

Use this dialog box to locate a specific event in the Event Viewer event list.

Event Types
Specifies the types of event to search for. Select the check box next to any or all of the following event types:

■ Information

■ Warning

■ Error

Event Source
Searches for events logged by the specified source. Select the name of a source in the list or click All.

User
Searches for events associated with the specified user. Select the name of a user in the list or click All.

Search Direction
Specifies the direction of the search, beginning with the selected event. Select Up or Down.

Find Next

Selects an event in the event list that matches the specified criteria. Click to search for the next event that matches the specified criteria.

**Related procedures:**

■ Finding a Specific Event in the Event Viewer (page 1630)

# Choose Columns Dialog Box

Use this dialog box to control the columns that are displayed in the event list of the Event Viewer.

Hidden Columns

Displays a list of the columns currently not displayed in the event list.

Add

Adds a column to the event list. Select the column name in the Hidden Columns list and click Add.

Remove

Removes a column from the event list. Select the column name in the Displayed Columns list and click Remove.

**NOTE** The Type column cannot be removed.

Displayed Columns

Displays a list of the columns that are currently displayed in the event list.

**Related procedures:**

■ Controlling the Columns Displayed in the Event Viewer (page 1627)

# Multi-View Block Definition Properties Dialog Box

Use this dialog box to create new multi-view blocks and edit existing multi-view blocks in the current drawing.

Multi-view Block Name

Displays the name of the multi-view block currently being edited or created. The list displays the names of all multi-view blocks defined in the drawing.

Edit

Displays the Edit Multi-View Block Definition dialog box, which allows you to edit the name of a multi-view block definition.

New

Displays the New Multi-View Block Definition dialog box, which allows you to specify a name for a new multi-view block definition.

Multi-view Block Description

Specifies a description for the multi-view block.

**Multi-view block components**

2D View Blocks

When selected, the view blocks list displays the names of the blocks that are shown when the view direction is a plan view.

3D View Blocks

When selected, the view blocks list displays the names of the blocks that are shown when the view direction is a 3D view (any direction other than plan).

Add

Displays the Blocks dialog box, which you can use to select blocks for the current multi-view block definition. The selected blocks are added to the 2D view block list or 3D view block list depending on the state of 2D view blocks and 3D view blocks buttons.

Remove

Removes the blocks selected in the view blocks list from the multi-view block definition. It is valid to remove all blocks from the list box control. No confirmation dialog box is displayed when the remove button is selected.

### Multi-view block preview

2D View

Sets the preview pane to plan view, and performs a zoom to extents to display all of the 2D blocks that define the multi-view block.

3D View

Sets the preview pane to a South-West isometric view, and performs a zoom to extents to display all of the 3D blocks that define the multi-view block.

Preview Pane

Provides a preview of the current multi-view block definition.

**Related procedures:**

■ Understanding Multi-View Blocks (page 1632)


# Multi-View Block Definition Properties Dialog Box

Use this dialog box to edit existing multi-view blocks in the current drawing.


# General Tab (Multi-View Block Definition Properties Dialog Box)

Use this tab to edit the name and description of the multi-view block.

Name

Specifies the name of the block.

Description

Specifies the description of the block.

This Style May Act As a Boundary For Automatic Spaces

This setting is not used by AutoCAD Civil.

Notes

Attaches notes (page 2451) to the block.

Floating Viewer

Displays a viewer dialog box where you can preview the appearance of the block.

# View Blocks Tab (Multi-View Block Definition Properties Dialog Box)

Use this tab to connect multi-view blocks to the view directions of individual display representations. To ensure that the multi-view block is visible in all views, assign a block to every view direction.

Display Representations

Specifies which display representation will use the view block settings that you specify. Select General to make assignments for plan views and Model for other model space views.

View Blocks

Specifies the block to use for each display representation. Click Add, and select a view block for the display representation. With the block selected, specify the View Directions that correspond to the selected block (see below).

View Directions

Specifies the view directions that correspond to the selected block. For example, to connect the view block to the left side of the multi-view block definition, select Left and clear all other directions.

You can connect a view block to multiple view directions. For example, if you have a symmetrical element that looks identical from left and right, you can select Left and Right.

The Other view direction represents a view that displays the block three-dimensionally. It is used for non-orthogonal (isometric or perspective) views of the custom object.

To ensure that the multi-view block is visible in all views, assign a block to every view direction.

Set Interference Block

Specifies a cut body for the multi-view block when the block is used as interference in other objects.

# Classifications Tab (Multi-View Block Definition Properties Dialog Box)

This tab is not used by AutoCAD Civil.

# Edit Multi-View Block Definition Dialog Box

Use this dialog box edit the name for an existing multi-view block definition.

Multi-view Block Name

Rename an existing multi-view block definition.

**Related procedures:**

■ Understanding Multi-View Blocks (page 1632)

# New Multi-View Block Definition Dialog Box

Use this dialog box to enter the name for a new multi-view block definition.

Multi-view Block Name

Provide a name for a new multi-view block definition.

**Related procedures:**

■ Understanding Multi-View Blocks (page 1632)

# Blocks Dialog Box

Use this dialog box to select blocks for the current multi-view block definition properties.

Blocks

Displays the names of all blocks in the current drawing. Select the blocks to add to the multi-view block definition. Multiple selection is supported.

**Related procedures:**

■ Understanding Multi-View Blocks (page 1632)

# Inquiry Tool Dialog Box

Use the Inquiry Tool to run inquiries on AutoCAD Civil objects.

Select An Inquiry Type

Specifies the inquiry to run. After you select an inquiry type, you are prompted to select the objects needed to run the inquiry.
Inquiry results are displayed in the Results area.

After the prompts are complete, you can click ![icon] or ![icon] in the value rows to change any of the selections you made. Some value rows also have lists you can use to change the selection.
For more information about the inquiry types, see Inquiry Types (page 1620).

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**NOTE** To run Corridor queries, a corridor section view must be active. Click Corridor menu ➤ View/Edit Corridor Section and then select the station. Use the Zoom command as needed to zoom closer to the view.

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![icon] Copy To Clipboard

Copies the inquiry results to the clipboard so you can paste it into other applications, such as Microsoft© Excel.

![icon] Copy To Text Screen

When selected, continually sends the inquiry data to the AutoCAD text window.

# Coordinate Tracker Dialog Box

Use this dialog box to track and capture X, Y, and Z drawing coordinates. The precision of the displayed value is determined by the coordinate precision drawing setting.

**X, Y, Z**

X:

Displays the X coordinate of the cursor location in the drawing.

Y:

Displays the Y coordinate of the cursor location in the drawing.

Z:

Displays the Z coordinate (elevation) of the cursor location in the drawing on the specified surface.

![icon] Continue Tracking

Resumes coordinate tracking after it has been suspended due to the entry of the coordinate capture keys.

![icon] Send XY

Sends the displayed X and Y values to the command line for use by the active command.

![icon] Send Z

Sends the displayed Z value to the command line for use by the active command.

| Send XYZ

Sends the displayed X, Y, and Z values to the command line for use by the active command.
For more information, see Sending Coordinate Tracker Values to the Current Command (page 1624).

### Surface

Surface

Displays the name of the surface from which the elevation value is read. Select a surface from the list. All surfaces in the active drawing are listed. If the drawing contains only one surface, that surface is automatically used to calculate Z values.

Surface Selector

Selects a surface in the drawing.

### Coordinate Capture Keys

Coordinate Capture Keys

Displays the keys that suspend coordinate tracking and capture the current coordinates. When the keys are pressed and held down, one by one in the specified order until all keys are pressed, cursor tracking is suspended and the X, Y, and Z values are captured at the current location of the cursor. You can then edit the coordinate values.

Coordinate Capture Keys Selector

Changes the coordinate capture keys. Enter the new coordinate capture keys. For more information, see Changing the Coordinate Tracker Coordinate Capture Keys (page 1625).

**Related procedures:**

■ The Coordinate Tracker (page 1623)

# Export To SDF Dialog Box

Use this dialog box to save a AutoCAD Civil drawing in Spatial Data Format (SDF). You can use an SDF file in AutoCAD Map 3D and MapGuide Enterprise.

The AutoCAD Civil alignment, point, parcel, and pipe network data is converted to AutoCAD objects with attributes. For more information, see Exporting Civil Data to SDF Files (page 1660).

SDF File Name

Specifies the file name. Defaults to the last file name used.

### Select Coordinate System

If the drawing already has a coordinate system specified, it is used automatically and these controls are not available.

Categories

Specifies the geographic zone.

Available Coordinate Systems

Specifies the coordinate system within the selected zone category. If you do not select a zone category, then the default (No Datum, No Projection) is enabled. If you select a zone category but do not select a coordinate system, then, by default, the first entry in the list is selected.

Selected Coordinate System Code

Displays the coordinate system code.

Description

Displays a description of the selected zone.

Projection

Displays the projection method for the coordinate system.

Datum

Displays the abbreviation for the datum on which the coordinate system is based.

**Related procedures:**

■ Exporting Civil Data to SDF Files (page 1660)

# DWF Publishing Options Dialog Box

Use this dialog box to access AutoCAD Civil-related settings for publishing 3D DWF data.

You can specify the objects to publish to the DWF file, how they are organized, and if the object properties are included in the DWF file.

## Object To Publish

All Model Space Objects

Specifies that all the objects in the drawing model space are published to the DWF file.

Selected Model Space Objects

Specifies that the selected objects in the drawing model space are published to the DWF file. Use the object picker button to select objects from the drawing.

## 3D DWF Organization

Specifies how to organize the objects in the DWF file.

Group By Xref Hierarchy

Specifies that objects in the DWF are arranged by Xref hierarchy in the viewer. If unchecked, externally referenced files are listed in the viewer like other objects.

Group Individual Objects By

Specifies how to group objects in the Model panel of the DWF Viewer. You can group them either by object type and style, or by the layer that the objects reside on.

## Options

Specifies additional DWF publishing options.

Publish With Materials

Specifies that the model will be exported with associated textures to various objects (for example, surfaces, pipes, or AutoCAD solids).
If you've assigned texture mapped materials to your models, those materials can be published with the 3D DWF file. Texture map orientation and scaling that is set in the drawing editor is retained in the published 3D DWF file.
There are some limitations to materials publishing.

■ The Diffuse Map channel is the only mapping that gets published. If you use Opacity, Reflection, or Bump maps in your material, they do not get published.

■ Procedural materials such as Wood or Marble do not get published.

**NOTE** Because the DWF Viewer and the rendering engine are different, you may encounter some deviation in how the texture maps appear in the DWF Viewer.

Include Properties From Objects

Specifies that the object properties will be available in the DWF file. The properties vary from object to object and include common properties such as Name, Description, Date Created, Created By, Date Modified, Modified By, Object Style, Layer, and Color. Each object has special properties as well. For example, an alignment includes start station, end station, and length.

**Related procedures:**

■ Exporting a Drawing to a 3D DWF File (page 1662)

# Glossary

This glossary contains terms relevant to AutoCAD Civil. Please see the AutoCAD glossary for additional platform terms.

**2D polyline**  A polyline with all vertices at the same elevation.

**3D face** A 3D face is an AutoCAD object that represents the surface of a 3- or 4-sided area, with each vertex potentially at a different elevation. You can view TINs (Triangulated Irregular Networks) as 3D faces. Using the SHADE command, you can shade 3D faces. Using the RENDER command, you can render the 3D faces. *See also* TIN (page 2485).

**3D polyline** A polyline with vertices at varying elevations.

**A** Grade Change.

**A.A.S.H.T.O.** American Association of State, Highway, and Transportation Officials.

**acre** A measure of land: 160 square rods; 4,840 square yards; 43,560 square feet.

**adjustment** A process that removes inconsistencies from the mathematical model of measured observations such as angles and distances.

**alignment** A series of 2D coordinates (northings and eastings), connected by lines, curves, or spirals, used to represent features such as the road centerlines, edges of pavement, sidewalks, or rights-of-way.

**angle** The difference in direction between two convergent lines measured in the units of degrees, radians, or grads.

**area** The quantity of plane space in a horizontal plane enclosed by the boundary of any polygonal figure.

**assembly** An AutoCAD Civil drawing object (AECCAssembly) that manages a collection of subassembly components, such as travel lanes, curbs, shoulders, and ditches, to form the structural elements of a roadway or other corridor-type structure.

**assembly set** The set of assemblies that are specified (referenced) by an assembly set file, used during intersection object creation (intersection design).

**astronomic azimuth** An azimuth derived from sunshots or starshots.

**azimuth** A clockwise angle measured from a reference meridian. Also known as north azimuth. It can range from 0 to 360 degrees. A negative azimuth is converted to a clockwise value.

**backsight** A previously established point used as a direction reference to another point. A survey instrument movements typically are locked to an angle of 0 degrees, and the vertical crosshair is set on the backsight. All subsequent shots are then taken by turning the instrument and the upper movement together, while the lower movement stays set on the backsight. All angles measured are, therefore, relative to the backsight.

**balance line** The line in a mass haul diagram at which cut and fill balance. *See also* mass haul line (page 2478).

**balance point** The point at which the mass haul line crosses the balance line (the station at which the cut volume and fill volume are equal). *See also* balance line (page 2468), mass haul line (page 2478).

**base surface** An existing ground or undeveloped terrain. Specified when creating volume surfaces. *See also* comparison surface (page 2470).

**baseline** *See* footprint (page 2474).

**BC:L** Curve Beginning.

**bearing** An angle measured from North or South, whichever is nearest, with the added designation of East or West. The angle is always less than 90 degrees (PI/2 radians or 100 grads) and is referenced by a quadrant number.

**BFS** Begin Full Super.

**BNC** Begin Normal Crown.

**BNS** Begin Normal Shoulder.

**BOA** Begin of Alignment.

**border** The visible limits of a surface. The border can be defined from a boundary or can be the result of a defined boundary and operations performed on a surface, such as a Delete Line operation (a hole is created in the surface). If a boundary is not defined on a surface, the exterior border is always defined as the extents of the surface triangulation.

**borrow pit** A pit that is created at a station along an alignment, to provide material to be used as fill in a mass haul diagram. *See also* dump site (page 2473).

**boundaries** Three kinds of closed polylines that limit the display area of the digital terrain model. Most common are outer surface boundaries constructed just outside the extremities of the dataset, eliminating unwanted interpolations across empty space where the surface has a concave shape. The following two types of internal surface boundaries are also used: hide boundaries, to punch holes in a surface (for example, a building footprint), or show boundaries, to create smaller surfaces by eliminating areas that fall outside the boundary.

**bounded volume** A method of calculating volumes using an existing AutoCAD object (for example, a polyline or polygon) to calculate the cut, fill, and net volume for the area bounded by the object.

**BP** Alignment Beginning.

**BP:STA** Alignment Beginning Point.

**BREAK** Grade break.

**breakline** A line used to connect the data representing a distinct surface feature, like a ridge line, edge of pavement, toe of a slope, centerline of a road, or flowline of a ditch or stream. When a breakline is defined, the surface triangulation must first follow the breakline, by placing triangle edges coincident with the breakline segments. This ensures the feature in the model is accurately depicted. Then, the rest of the interpolation is performed based on proximity. Breaklines are typically critical to creating an accurate surface model. It is the interpolation of the data, not just the data itself, that determines the shape of the model. *See also* non-destructive breakline (page 2478).

**breakline point** A point that is included in the defined breakline's list of vertices.

**BVC** Vertical Tangent-Curve Intersect.

**BVCE** Vertical Tangent-Curve Intersect Elevation.

**BVCS** Vertical Tangent-Curve Intersect Station.

**BVP** Profile Start.

**ByBlock** A setting specifying that a component of an object inherits the color or linetype associated with the object, or block, that contains it.

**ByLayer** A setting specifying that an object or component of an object inherits the color or linetype associated with the assigned layer.

**bypass target** A part in a pipe network that has been identified by the user to act as the target for any bypass flows to be directed to.

**catchment area** The area tributary to a lake, stream, or drainage system.

**center marker** A diamond-shaped graphic that marks the location of the centroid of a grading object. Right-click the center marker to access the Grading shortcut menu.

**chord** A straight line connecting two points on a curve: the Point of Curvature (PC) and Point of Tangency (PT). The curve joins with a line or another curve at these points.

**closure** The relative position of a traverse station as it compares to the same station position determined by a different set of observations or route of survey.

**clothoid spiral** A spiral in which the curvature is a linear function of the length of the spiral, so that the degree of curvature is zero when it meets the tangent and then increases to match the curvature of the adjacent curve. *See also* compound spiral, simple spiral.

**COGO** Coordinate Geometry.

**COGO points** The point objects that you create using the point-creation or point-import. COGO points are referred to simply as "points" in this documentation. The pieces of data associated with a point, such as point number, northing, and easting, are referred to as properties. *See also* properties (page 2480).

**collimation** The process of adjusting the line of sight or lens of an optical instrument so that it is properly located based on the other parts of the instrument.

**comparison surface** A proposed or existing terrain surface used in the creation of volume surfaces. *See also* base surface (page 2468), volume surface (page 2487).

**Compass rule** Corrections corresponding to the closing errors that assumes the closing errors are as much due to of errors in observed angles as errors in measured distances. Use the compass rule option when this is the case.

**composite volumes** A method of calculating volumes using top and bottom surfaces (a surface pair) to establish cut, fill, and net volume values.

**compound curve** A curve consisting of two or more arcs of different radii curving in the same direction, and having a common tangent or transition curve at their point of junction.

**compound spiral** A spiral that provides a smooth transition between two adjoining curves of different radii but in the same direction. It has a finite radius on either side/end. *See also* clothoid spiral (page 2470).

**confidence interval** In statistics, a region or area that has a known probability of containing a random sample. In surveying, a 95% confidence interval indicates that the surveyor can be 95% sure that a point or observation lies within the region or established parameters.

**construction geometry** Unconnected portions of parcel and alignment geometry. For example, if you use the tan-tan method and create a number of connected lines, the whole feature is solved and the display characteristics are controlled by the object style. If you create some connected lines and then add a line that is not connected, the result is a piece of 'construction' geometry.

**contour** A line that connects points of the same elevation or value relative to a specified reference datum.

**control point** Points with a unique identifier that are created at a known location (northing, easting, or latitude, longitude, with an optional elevation or description) and are not affected by adjustments or corrections. Control points are added to a survey database and are managed within a named survey network.

**control region** A region that is defined by applying grading criteria to a footprint. A footprint can have a single region along its entire length, or it can be subdivided into a number of control, transition, and void regions. *See also* footprint (page 2474).

**coordinates** Values that specify exactly where a point is in space in terms of three planes: X, Y, and Z (easting, northing, and elevation).

**corridor** Any path, the length and location of which are typically governed by one or more horizontal and vertical alignments. Examples are roadways, railways, traveled ways, channels, ditches, utility runs, and airport runways.

**corridor feature line** A component of a corridor object. Created by joining subassembly point codes from one station to another.

**corridor surface** A component of a corridor object. Surface triangulation is created from a corridor subassembly point and link codes.

**Crandall rule** A method of balancing a traverse in which all the angular error is distributed throughout the traverse and all adjustments to the traverse result from modifying the traverse distances. The modification distance made to each leg is such that the sum of the squares is a minimum.

Corrections correspond to the closing errors, which assumes the closing errors are random and normally distributed. It assumes all the angular error has been adjusted before the adjustment routine.

**crest curve** In a profile, a vertical curve on the crest of a hill or similar location where the grade leading into the curve is greater than the grade leading out of the curve. In a crest curve, the point of vertical intersection (PVI) for the tangents is above the curve. *See also* sag curve (page 2482).

**criteria-based design** The process of associating agency-specific standards to an alignment or profile. Agency standards, which are typically based on superelevation and design speed requirements, are contained in a customizable design criteria file.

When an alignment or profile object is created, standards from the design criteria file can be associated with the alignment or profile to ensure that the horizontal and vertical curves comply with the minimum standards. User-defined design checks can be used to identify and report standards violations. *See also* design criteria file (page 2472).

**cross section** *See* section (page 2482).

**CS** Curve-Spiral Intersect. *See* curve to spiral (page 2471).

**CS_LRB:L** Compound Spiral Large Radius at Beginning.

**CS_LRE:STA** Compound Spiral Large Radius at End.

**CS_SRB:L** Compound Spiral Small Radius at Beginning.

**CS_SRE:STA** Compound Spiral Small Radius at End.

**curb return alignment** An alignment that connects the edges of two intersecting roadways. The most common curb return geometry is a simple circular fillet. In a typical intersection, curb return alignments are placed in each of the four quadrants, between edges of pavement.

**curve to spiral** A point on a horizontal alignment where a curve meets a spiral.

**cut slope** The slope created when the footprint falls below the existing ground line. The resulting slope matching up into the existing ground is called a cut slope because the existing ground must be cut (removed) during construction.

**data band** A graphic frame that is associated with a profile view object or section view object. The data band contains annotations for the profile or section view, as well as for the parent horizontal alignment. Some common annotations include elevation data, stations, and cut/fill depths.

**data collector** A device surveyors use to automatically record the observation data they collect in the field. They then download and convert the raw survey data to a field book file, which is written in the Survey Command Language format, or a LandXML file, which uses the LandXML schema to describe the Survey data. Surveyors can then import this file to create points in the database and figures in the drawing.

**data reference** A read-only copy of an object from another drawing. The referenced object can be used in multiple drawings and stylized locally. After the official copy of the object is updated, drawings with references to it are notified of the opportunity to update their local copy. There are two data reference types used in AutoCAD Civil: data shortcuts and Vault references. Both types can reference surfaces, alignments, profiles, pipe networks, and view frame groups.

**data shortcut** An object that can create a data reference between drawings in a project. Data shortcuts are not used with Vault projects.

**datum** A reference value. All elevations or coordinates are set relative to this value. In surveying, you can use two datums (horizontal and vertical).

For global coordinate systems, a datum refers to the ellipsoid information and the techniques used to determine positions on the Earth's surface. An ellipsoid is part of a datum definition.

**daylight line** A line showing the line of zero cut or fill within the job area. For grading objects, it represents the target line produced by grading to a specified surface, distance, or elevation.

**decimal degree** The measure of an angle in decimal units. For example, 3°30'36" equals 3.51 decimal degrees.

**Decreasing** Station Equation Decreasing.

**definition list** A list that contains all the operations performed on a surface. By turning the items in the list on and off, you can modify the surface, return it to a previous state, and so on.

**deflection angle** A horizontal angle measured from an extension of the preceding line, right or left.

**degrees, minutes, seconds** (DMS) A representation of an angle in degrees, minutes, and seconds in which a full circle contains 360 degrees, each degree 60 minutes, and each minute 60 seconds. A typical bearing in DMS measurement looks like: N45°45'58"E. Using this format, 3°30'36" is entered as 3.3036.

**Delaunay triangulation** A calculation method used in the creation of TIN networks. Given a set of data points, Delaunay triangulation produces a set of lines connecting each point to its natural neighbors.

**DEM** (Digital Elevation Model) An array of elevations taken on a regularly spaced horizontal grid.

**description keys** A method of translating descriptions to help standardize point data when a variety of data sources are used. For example, descriptions of EROAD, EPAVE, ERD, and EDGEROAD can all be translated to a description of EOP. The layer, point style, and point label style options can greatly enhance automatic base plan generation and the overall organization of the drawing.

**design check** A user-defined expression used to verify that an entity meets the minimum design standards that were established for the alignment or profile object. Design checks may be defined for different entity types, such as lines, curves, and spirals. A design check must be saved in a design check set to be applied to an alignment or profile. *See also* design check set (page 2472).

**design check set** A user-defined collection of commonly used design checks. You specify a design check set either during alignment or profile creation, or after creation using the object properties dialog box. *See also* design check (page 2472).

**design criteria file** A file that contains minimum design standards for alignment and profile objects. The design criteria file may be customized to support local design standards for design speed, superelevation, and minimum speed, radius, and length of individual entities. *See also* criteria-based design (page 2471).

**design rule**  A set of automatic constraints for some objects in the drawing that are enforced by applying a specific style. For example, if you are laying out alignments, you can specify a rule for a curve radius.

**drawing objects** Objects that exist in a single drawing. *See also* project objects (page 2480).

**dump site** A site that is created at a station along an alignment, at which cut material removed from a mass haul diagram is dumped. *See also* borrow pit (page 2469).

**easting** A linear distance eastwards from the North-South line which passes through the origin of a grid. Equivalent to the X coordinate in an XYZ coordinate system.

**EC:STA** Curve End.

**EDM** Electronic distance measuring devices that measure the round-trip transit time of a pulsed signal from which distance is computed.

**EDM Offset** The vertical distance from the scope center to the EDM center. AutoCAD Civil uses this value to calculate distance.

**EFS** End Full Super.

**EGL** Energy Grade Line.

**elevation** The vertical distance from a datum to a point or object on the Earth's surface. The datum is considered to be at sea level. Equivalent to the Z coordinate in an XYZ coordinate system.

**elevation point** A point that marks an elevation change but does not break the horizontal geometry.

**ENC** End Normal Crown.

**energy grade line** A line in a pipe network or storm sewer drainage system that represents the total available energy in the system (potential energy, or static head, plus kinetic energy, or velocity head).

**ENS** End Normal Shoulder.

**EOA** End of Alignment.

**EP** Alignment End.

**EP:STA** Alignment End Point.

**error ellipse** An elliptical region that shows the confidence interval of an adjusted point. For example, if the confidence is set to 99 %, you can be 99 % sure the calculated ellipse contains the true location of the point.

**EVC** Vertical Curve-Tangent Intersect.

**EVCE** Vertical Curve-Tangent Intersect Elevation.

**EVCS** Vertical Curve-Tangent Intersect Station.

**EVP** Profile End.

**face** A three-dimensional surface triangle. A face is represented by either a 3D face object or 3D line objects.

**face 1** An angle measurement in which the scope is not flopped. Also called direct.

**face 2** An angle measurement in which the scope is flopped. Also called inverse or indirect.

**feature line** An object in that the grading commands can recognize and use as a footprint. Usually, a line that marks some important feature in the drawing, such as a ridge line, or the bottom of a swale. *See also* footprint (page 2474).

**field book** The permanent detailed record a surveyor makes of all observations made in the field. In AutoCAD Civil, a field book file, (*.fbk), it can be used as a source of survey data.

**field code** Syntax that corresponds to a predefined convention in a surveying application. Field codes are assigned to each survey point to automate the assignment of point and line properties and to determine the linework connectivity.

**Field to Finish** A process of determining the line and curve connectivity between points surveyed in the field that have the same feature name within a point code. This process also assigns symbols to point features based on a feature name within a point code.

**figure** Special linework automatically created in a drawing when you import a field book containing figure data. Also created when using AutoCAD Civil survey figure commands. You can control the layers for figures by using figure prefixes.

**figure prefix** Used to automatically place figures on user-defined layers based on the beginning part of the figure name. For example, the figure prefix EP can be used to place the figures EP, EP1, EP2, and EPL all on the same layer. Using figure prefixes in conjunction with description keys can largely automate the process of constructing a well organized base plan.

**fill slope** The slope created when the footprint falls above the existing ground line. The resulting slope matching down into the existing ground is called a fill slope because material must be brought in to fill the area during construction.

**fixed entity** An alignment entity with a fixed position, not necessarily tangent to another entity for the definition of its geometry. *See also* floating entity (page 2474), free entity (page 2474).

**floating entity** An alignment entity that is tangent to one other entity (before or after) for the definition of its geometry. *See also* fixed entity (page 2474), free entity (page 2474).

**flow rate** A hydraulic property, often used in desigining storm sewer and other systems, that represents the volume of fluid over time that flows through a system.

**footprint** The object to which grading criteria are applied. A footprint can be an open or closed 2D or 3D geometric figure that is a feature line, parcel line, or survey figure. *See also* feature line (page 2473).

**footprint vertices** The endpoints that define the segments of a footprint with an XYZ location and stationing starting at the first vertex.

**foresight** A point to which an instrument sighting is made to measure or establish its elevation or horizontal position.

**free entity** An entity that is tangent to two other entities (before and after) for the definition of its geometry. *See also* fixed entity (page 2474), floating entity (page 2474).

**free haul** The material moved within a certain defined distance of its point of excavation (the free haul distance), free of charge by the contractor. *See also* mass haul (page 2477), overhaul (page 2479).

**front yard** The offset distance from the parcel frontage segments to the setback line.

**frontage** The parcel segments that are adjacent to a right-of-way. The frontage is also considered to be the parcel segments located at the front of the parcel.

**frontage offset** A user-defined length used to offset the current frontage definition to the inside of the parcel being subdivided. When the frontage offset is used, the minimum frontage length is applied at and along the frontage offset instead of the frontage.

**full description** The expanded description of a point after description key matching has taken place.

**geodesic** On a surface, the shortest line between two points, either a line or curve from one point along an ellipsoid to another.

**geodetic** A basic relationship to the Earth that takes into account the curvature of the Earth's sea level surface. For example, a geodetic distance is a distance or angle in which the Earth's curvature is taken into account, versus a distance or angle measured on a flat paper map.

**grade** A method of reporting ground inclination in which the change in elevation is expressed as a percentage of the horizontal distance travelled. For example, if the ground rises one linear unit (meter or foot) over a horizontal distance of five units, the grade is 20%. *See also* slope (page 2482).

**grade line** *See* layout profile (page 2476).

**grade point** A point in a mass haul diagram at which mass haul transitions from cut to fill. When free haul is measured from grade points, the highest point (or lowest, if below the balance line) is the grade point. *See also* balance line (page 2468), mass haul line (page 2478).

**grading** The process used to model the finished ground surface.

**grading criteria** Parameters such as target and projection method for the grading.

**grading face** The area bounded by the footprint, the daylight line, and the two projection lines.

**grading group** A collection that ties grading objects together for the purposes of surface creation and volume calculation.

**grading target** The grading target defines what the projection lines from the footprint will intercept. The three choices for targets are: surface, relative or absolute elevation, and distance. *See also* daylight line (page 2472).

**grads** A system of angle measurement in which one grad equals 1/100 of a 90° angle, or 360° = 400 grads.

**grid** A system of lines parallel to a given set of axes at a specific spacing. Grids are used to visualize surfaces and calculate volumes. A grid is also used for geodetic purposes.

**grid distance** The distance between two points based on a coordinate zone, not on local northing and easting coordinates.

**grid easting** The easting coordinate that is based on a selected coordinate zone, as opposed to the local easting, which is based on the surveyor's base point. *See also* local easting (page 2477).

**grid northing** The northing coordinate that is based on a selected coordinate zone, as opposed to the local northing, which is based on the surveyor's base point. *See also* local northing (page 2477).

**grid surface** A type of surface created from a DEM or SDTS file, or imported from a LandXML file. A grid surface consists of a sampled array of elevations for a number of ground positions at regularly spaced intervals.

**grid volume surface** A differential grid surface based on user-specified base and comparison surfaces. The grid method of volume calculation measures the difference in elevation between two surfaces at each intersection in a user-defined grid.

**grip** A moveable point on an object that you can drag to edit the object dynamically.

**handle** The unique identifier of a drawing object, used as a reference within the software. Users normally do not need to know about handles unless they are using the Data Shortcut Editor.

**hectare** A measure of area, generally relating to land, of 10,000 square meters or approximately 2.47 acres.

**HGL** Hydraulic Grade Line.

**High-definition surveying (HDS)** 3D laser scanning typically associated with terrestrial based laser scanners. The terms "High-Definition" and "HDS" are trademarks of Leica Geosystems.

**horizon** An axis that the scope of a theodolite or transit rotates about when moved vertically. The axis of rotation perpendicular to the vertical axis.

**HP** High Point.

**hydraulic grade line** A line in a pipe network or storm sewer drainage system that represents the elevation head and pressure head of fluid at any point along a system.

**import event** A named collection in a survey database that provides a context to the specific data that was imported using any of the Survey Import commands, such as the Import Survey LandXML command, or the Import Survey Data wizard.

**Increasing** Station Equation Increasing.

**incurve** A spiral with a curve radius that decreases along the route of the defined direction.

**interference checking** A AutoCAD Civil feature available with pipe networks that lets you identify pipe network parts (pipes and or structures) that physically overlap each other, or are too close to one another based on predefined proximity criteria.

**intersection** The point where two or more lines, arcs, figures, or objects join or cross in two- or three-dimensional space.

**inverse** An inverse calculation on a closed figure calculates the bearings and distances between coordinates and reports the area. Because you use exact numbers (coordinates) to determine corners, no closure error is reported.

**junction loss** Or junction loss coefficient. A hydraulic property value used in storm sewer drainage systems that can be computed and defined in a pipe network. It is associated with the loss of energy through a junction. This numeric value has no units associated with it, and can be automatically calculated or manually entered using the Hydraflow Storm Sewers extension application. It is used to compute minor loss for hydraulic analysis.

**K** Curve Coefficient.

**known capacity** A numeric value that represents the flow in a pipe network or storm sewer drainage system as designated by the user.

**Kriging** A method of surface smoothing that uses known values and a semivariogram to determine unknown values. Based on the semivariogram used, optimal weights are assigned to known values to calculate unknown ones. Since the variogram changes with distance, the weights depend on the known sample distribution.

**L** Left.

**label component** Text, a block, direction arrow, line, or tick that is used to construct a label style. Label styles can be made up of multiple label components.

**label set** A collection of label definitions for multiple label types, such as station labels and geometry point labels. For example, alignment station labeling can be composed of major station labels, minor station labels, and geometry point labels.

**latitude** The angular distance measured on a meridian north or south from the equator.

**layout profile** A profile object that represents the finished ground elevations along a horizontal alignment or other linear feature that supports profile views. Typically, this data is designed, not derived from a pre-existing source, and consists of a series of vertical tangents connected by vertical curves located at each point of vertical intersection (PVI). This profile is also known as vertical alignment, profile grade line, grade line, and finished grade profile.

**layout toolbar** A floating, dockable toolbar that groups object-specific design and editing tools.

**LB:L** Line Beginning.

**LC** Level Crown.

**LE:STA** Line End.

**least squares** A method of balancing a traverse in which the squares of the differences between the unadjusted and adjusted measurements (angles and distances) are summed and reduced to a minimum. This method uses the error specifications in the current equipment settings to determine the expected source of errors, and weights the individual measurements accordingly.

**length weighted distribution** A vertical adjustment that distributes the vertical closing error to each line, at the same ratio as the length of that line is to the total length of the traverse (similar to the Compass rule).

**LIDAR** Light Detection and Ranging. Typically associated with airborne laser scanning.

**linework code set** A file that dictates the syntax of linework connectivity commands that are specified within a field code. A feature that is an edge of pavement may be abbreviated as EP or EOP in the field code but the set of abbreviations are part of the field coding standard established by a company. The linework code accompanies the feature name within the field code, for example "EP B", where "EP" is the feature name, and "B" is the linework code to begin a survey figure.

**link codes** A set of standard codes that define the display and behavior of links used in subassemblies.

**local copy** A copy of a project drawing that resides in your local folder, as opposed to the master copy of the file in the project Vault database. Also refers to a copy of a project point that exists in a local drawing.

**local easting** The easting coordinate that is based on the surveyor's assumed horizontal base point, as opposed to the grid easting, which is based on the global coordinate zone. *See also* grid easting (page 2475).

**local elevation** The elevation coordinate based on the surveyor's assumed vertical base point, or benchmark, versus a real world elevation value.

**local northing** The northing coordinate that is based on the surveyor's assumed horizontal base point, as opposed to the grid northing, which is based on the global coordinate zone. *See also* grid northing (page 2475).

**locked point** A drawing point whose coordinate data cannot be modified. Point locking applies only to the drawing in which the point resides. *See also* protected point (page 2481).

**logical name mapping** The process of mapping logical names in subassemblies to actual AutoCAD Civil object names.

**longitude** The angle between the plane of a given meridian and the plane of the Greenwich meridian.

**LP** Low Point.

**LSM** Low Shoulder Match.

**MAN** Manual.

**map check** A map check reports how accurate your angle observations are, and the area of closed figures. You can perform a map check on an open or closed figure that was drawn by using angles. Because the accuracy of the figure is based on the precision of your angle measurements, an amount of closure error is associated with such a figure. The greater the precision you use, the less error there will be. In contrast, if you draw a figure between known coordinates, then there is no need to perform a map check, because there will be no error.

A map check reports precision and mathematical closure based either on bearing and distance entries of a closed figure, or for an open figure between two known points. The accuracy of a map check is based on the precision used to calculate it. For example, if you use data that is rounded to the nearest foot, then the results are less accurate than if you use data rounded to the nearest one-hundredth of a foot.

**masking** A process of obscuring objects so that text can be placed over them in a clear area. Masking maintains legible text without destroying the objects underneath.

**masks** Polygons used to mask the visible portions of a surface.

**mass haul** The volume of excavated material times the distance it is required to be moved. A mass haul diagram presents a visual representation of the cumulative cut and fill material volumes on a project. Mass haul diagrams (along with grading summaries) are the contractor's primary tools for bidding earthwork. *See also* free haul (page 2474), overhaul (page 2479).

**mass haul line** A line marking the balance between cut and fill in a mass haul diagram. When the mass haul line is above a balance line, it indicates material to be cut. When the mass haul line is below the balance line, it indicates a volume to be filled. *See also* balance line (page 2468), mass haul (page 2477).

**match line** *See* daylight line (page 2472).

**mid-ordinate** On a circular arc, the distance from the midpoint of a chord to the midpoint of the subtended arc.

**modifier** A formula for formatting property field values, such as decimal precision for an area. You can use predefined modifiers, such as acres, feet, and meters, or you can define a custom modifier. *See also* property fields (page 2481).

**nadir angle** The nadir angle is opposite the zenith.



**Natural Neighbor Interpolation (NNI)** A method of surface smoothing supported on TIN surfaces. NNI uses Delaunay triangulation to determine the elevation of an arbitrary point based on the elevations of known neighbor points.

**network** A series of interconnected lines that represent the observed instrument setups.

**non-control point** Points that are created with a unique identifier, northing, easting (or Latitude, Longitude), and optional elevation and description. A non-control point represents a point whose location is determined from a reduced coordinate (N,E,Z) observation, such as point created by the NE SS survey command. A non-control point is not connected to other survey observations and remains unaffected by a network analysis, but still resides within the context of a survey network. Non-control points are added to the survey database and are managed with a named survey network. Non-control points can be promoted to control points if you reference the point as a control point for creating a traverse, or reference the point as a setup to make observations to other points that can affect locations during an analysis.

**non-destructive breakline** A breakline that is not crossed by triangulation lines in a TIN. Instead, new vertices are added to the breakline at the intersection of each TIN line and the breakline. The new points create additional surface triangles. This is useful when you do not want the elevation of a surface to be interpolated inside an area that you know to be a constant elevation.

**northing** A linear distance northwards from the East-West line which passes through the origin of a grid. Equivalent to the Y coordinate in an XYZ coordinate system.

**note reduction** The process of taking field measurements and converting them from 3-dimensional to 2-dimensional in order to use the measurements in a plan view of the drawing. For example, note reduction includes the process of converting vertical distances to horizontal distances.

**null assembly** A placeholder assembly that is used during intersection object creation, when a referenced assembly cannot be found. A null assembly generally contains no subassemblies other than a marked point. *See also* assembly set.

**object** In AutoCAD Civil, an element in a drawing, for example, a surface, that can maintain a relationship with other objects.

**object model** The underlying system of links and dependencies between objects. In the object model, changes in one object can be passed on automatically to all the objects associated with it.

**occupied point** A point in a traverse loop where the survey instrument is set up and observations are recorded.

**offset alignment** A dynamic alignment created at an offset distance from another alignment, such as a road edge offset from a centerline alignment. The offset alignment geometry cannot be edited directly, but it responds dynamically to edits of the parent alignment.

**outcurve** A spiral with a curve radius that increases along the route of the defined direction.

**overhaul** The excavated material that must be hauled beyond the free haul distance. *See also* free haul (page 2474), mass haul (page 2477).

**override** A value for a setting that replaces the value already set at the next higher level.

**Panorama** A window that displays data in table form for the objects in a collection that is selected in Toolspace. For example, if you select a point group, the Panorama table displays a row for each point.

**parcel** A discrete piece of 2D area. For example, a subdivision is composed of numerous parcels. Synonymous with lot.

**parcel node** A point where two or more parcel segment ends join.

**parcel segment** A parcel boundary element, a line, or a curve.

**Part Builder** A parametric modeling tool used to create and modify the set of pipe network parts (pipe and structure shapes) that are available in pipe network part catalogs.

**part catalog** An xml file that contains definitions for the three-dimensional, parametric shapes of pipe and structure objects used in pipe networks.

**parts list** A set of references to pipe network parts (pipes and structures) whose shapes are defined in a catalog called the part catalog.

**pass-through point** A point on the path of a line or curve, often used to define an alignment. A pass-through point on a curve can be used as a grip to control the position of the curve. However the alignment is edited, its geometry has to go through the pass-through point.

**passing sight distance** The distance measured to a point where an approaching vehicle comes into view ahead of a driver on an undivided road. This is used to calculate vertical crest curves.

**pay item** A specific unit of work for which a price is provided and paid to a contractor while a project is under construction.

**pay item list** A master list provided by the contracting authority, listing pay item numbers, item specifications, and item units of measure.

**PC** Tangent-Curve Intersect. *See* point of curvature (page 2480).

**PCC** Compound Curve-Curve Intersect.

**PI** Tangent-Tangent Intersect. *See* point of intersection (page 2480).

**pipe network** A pipe network object manages a collection of pipe objects and structure objects that are used to represent a pipe network in a drawing.

**pipe object** A type of object used to visually represent a pipe in a pipe network, such as a circular, rectangular, or egg-shaped pipe.

**plan readable** Text that you can read easily in a normal plan view, that is, placed at an angle less than 270 degrees and more than 90 degrees. Also called right-reading.

**plan view** The view of a site if you look straight down from an elevated position.

**point codes** A set of standard codes that define the display and behavior of points used in subassemblies.

**point group** Collection used to group the points in a drawing into smaller, more manageable units. For example, you can create a point group that contains all of the points in a drawing that meet certain elevation criteria.

**point list** The list of the points that belong to a point group.

**point marker** *See* point symbol (page 2480).

**point of curvature** (PC) The point where an arc is drawn from a tangent.

**point of intersection** (PI) The point where two tangents meet on a horizontal alignment. Curves and spirals also have points of intersection, which indicate where the tangents would meet if they were extended outward.

**point of tangency** (PT) The point where a curve meets a tangent.

**point of vertical intersection** (PVI) In a profile, the point where two tangent lines meet.

**point symbol** A point location marker. When you add points to a drawing, point symbols are created to represent the points. The point style referenced by a point describes how the point symbol is drawn.

**polyface** A 3-dimensional (polygon) mesh object. Each face is capable of having numerous vertices.

**PRC** Reverse Curve-Curve Intersect.

**prism** A faceted glass reflector used to return the signal from an EDM, whereby the EDM can determine the distance to its reflection point.

**prism constant** The distance between the point of plumb and the reflection point within the prism.

**prism offset** The vertical distance between the theodolite target point and the optical center of the reflector.

**profile** An object that contains elevation data along a horizontal alignment or other line. There are two main types of profiles: surface and layout. Profile data objects can be viewed within a profile view object.

**profile grade line** *See* layout profile (page 2476).

**profile view** An object that manages the graphic display of profile data objects within a drawing. A profile view is essentially a graph with two primary axes: the x-axis represents horizontal distance along the referenced horizontal alignment (or other linear feature). The y-axis represents elevations. Profile view objects can also include grid display components and data bands.

**project drawing** A drawing that has been added to a project.

**project objects** Objects that exist in a project drawing and have been designated as *shared*, which means that they can be accessed by others. *See also* drawing objects (page 2473).

**projected object** An object in plan view of a drawing that is projected into a profile view or section view. AutoCAD objects that can be projected include points, blocks, 3D solids, and polylines. AutoCAD Civil objects that can be projected include points, feature lines, and survey figures.

**projection lines** In a grading, the lines that designate face edges within a region for break points on the footprint or the daylight line, and for the facets of curves (corner cleanup, vertical curves).

**properties** The settings that apply to a particular instance of an object.

**property fields** The placeholders in labels that contain content, such as text or graphics, along with format modifiers unique to specific features. Property fields can be named and their values defined according to the feature with which they are associated. *See also* modifier (page 2478).

**Prospector tab** The part of Toolspace where you access drawing and project objects. Objects are arranged in a tree or hierarchy with folders and subfolders that you navigate in standard, Windows-Explorer fashion. *See also* Settings tab (page 2482).

**protected point** A project point that you cannot check out and therefore edit.

**proximity breakline** A breakline that is drawn as a polyline without snapping to points in the drawing. The northing, easting, and elevation of the breakline vertices are determined from the nearest point contained in the surface point data, after generating the surface.

**PT** Curve-Tangent Intersect. *See* point of tangency (page 2480).

**PVI** Point Of Vertical Intersection. *See* point of vertical intersection (page 2480).

**quadrant** One of the sections resulting from dividing a circle into four equal parts. Quadrant 1 is the NE corner, and quadrants 2, 3, and 4 proceed clockwise around the compass. Bearings are usually referenced by quadrant number.

**quantity takeoff** The analysis of detailed cut and fill requirements along an alignment. A quantity takeoff report provides information on the total volume of material required to create a finished grade surface, including the process of cutting existing ground and refilling it with a different material, such as coarse gravel.

**R** Right.

**radians** A system of measure in which 2 pi radians equals 360°.

**raw description** The original description of a point, before description key matching takes place. Often corresponds to the point description entered in the field by a surveyor.

**raw station** A station value on an alignment, either formatted or unformatted, that does not take into account any station equations applied to the alignment.

**RC** Reverse Crown.

**rear yard** The offset distance from the parcel rear segments to the setback line.

**reference object** Within a project drawing, the read-only geometry of an object that exists in another project drawing. Objects that can be referenced include surfaces, alignments, profiles, and pipe networks. The host drawing can apply local object styles and annotations, and perform some analysis on reference objects.

**Referenced Text** A label component that contains references to other AutoCAD Civil objects.

**region (grading)** The area where grading criteria are applied to a lot line or a feature line.

**region lines** The projection lines that designate the start and end of the grading regions (criteria or transition) by connecting the footprint to the daylight line.

**return period** A time period that represents an interval at which an event, such as a large storm, occurs. It is used to calculate various flow control methods in a pipe network or storm sewer drainage system.

**right-of-way** (ROW) The allowable work area for an alignment. Property lines of the property owners who reside adjacent to the construction site generally specify these limits, which are called right-of-way lines.

**right-of-way parcel** A parcel that is created from an alignment that crosses the original parent parcel.

**rod** A slender bar of wood or metal marked with some type of measurement used to measure elevation and/or distance.

**ROW** *See* right-of-way.

**sag curve** In a profile, a vertical curve at the bottom of a valley or similar location where the grade leading into the curve is less than the grade leading out of the curve. In a sag curve, the point of vertical intersection (PVI) for the tangents is below the curve.

**sample** The process of obtaining elevation information from an existing terrain model or surface.

**sample line** A line that typically cuts across an alignment, and that can be used for creating cross sections.

**SBO** Shoulder Breakover.

**SC** Spiral-Curve Intersect. *See* spiral to curve (page 2483).

**SDTS** *See* Spatial Data Transfer Standard (page 2482).

**section** An object that contains elevation data along a sample line.

**setback line** A closed polygon that is the result of offsetting and intersecting each parcel segment by its respective front/side/rear yard distance. The exterior of the setback line typically represents the area where permanent structures are not permitted.

**settings** A collection of properties and styles that apply to an object.

**Settings tab** The part of Toolspace where you access the styles for the different types of objects, including object labels and tables. *See also* Prospector tab (page 2481).

**setup** Instrument setups that are defined in the Survey database.

**shape codes** A set of standard codes that define the display and behavior of shapes used in subassemblies.

**shortcut** *See* data shortcut (page 2472).

**shortcut menu** A menu that is displayed when you select an object and right-click. Shortcut menus are context sensitive so that only commands that are relative to the object that you selected are displayed.

**side yard** The offset distance from the parcel side segments to the setback line.

**sideshot** A point that is created with a unique identifier, resulting from relative observations (such as angles, distances, vertical angles, and target heights) taken from a survey station or setup to locate a point that is not intended to be used as a base for the extension of the survey. For example, at one station (setup) in a traverse loop, you can survey points for stone walls, trees, buildings, and wetlands.

**simple spiral** A spiral where the large radius end has an infinite radius and the small radius end has a finite radius, therefore providing a smooth transition from a tangent (infinite radius end) to a curve (finite radius end). *See also* clothoid spiral (page 2470), compound spiral (page 2470).

**site** A collection of objects that are managed by means of common topology. The objects that participate in the topology are parcels, alignments, and grading. *See also* topology (page 2485).

**slope** A method of reporting surface inclination as a ratio that expresses the horizontal distance in which the elevation changes by one linear unit. For example, if the ground rises 3 units over a horizontal distance of 15 linear units (meters or feet), the slope is 5:1 (5 to 1). *See also* grade (page 2475).

**slope projection** The method of grading to a target that is either Slope (H:V value) or Grade (percentage value).

**south azimuth** Azimuths south of the equator are referenced to due South clockwise.

**Spatial Data Transfer Standard** A file format designed as a mechanism for the transfer of spatial data between various computer systems. The SDTS Format is designed to transfer data with complete content transfer (no loss of information).

**SPI** Reverse Spiral Tangent.

**spiral** *See* clothoid spiral (page 2470), compound spiral (page 2470), simple spiral (page 2482).

**spiral to curve** (SC) A point where a spiral meets a curve.

**spiral to tangent** (ST) A point where a spiral meets a tangent.

**split profile** A profile view that has a readjusted datum at a station where one or more profiles goes out of bounds. This condition happens if a profile view that has a user-defined height is not able to adequately portray the full extents of a profile.

**spot elevation** The elevation of a single point in the drawing. Used to define areas that are sparse in contour data when generating a TIN using contour information. Areas that may also need spot elevations are the top of hills, valleys, and bottom of swales.

**SS** Spiral-Spiral Intersect.

**SS_LRB:L** Simple Spiral Large Radius at Beginning.

**SS_LRE:STA** Simple Spiral Large Radius at End.

**SS_SRB:L** Simple Spiral Small Radius at Beginning.

**SS_SRE:STA** Simple Spiral Small Radius at End.

**ST** Spiral to Tangent Interesct. *See* spiral to tangent (page 2483).

**stacked profile views** A collection of related profiles drawn in separate, vertically arranged profile views. Typically, a centerline profile is contained in one profile view, and its left and right offsets are drawn in profile views that are placed above and below the centerline profile view.

**stacked text** When you drag a label from its point of origin, label text can be rearranged by specifying settings for text justification, text height, and relationship to borders.

**stadia** A technique of distance measurement using a rod and a stadia transit.

**stakeout** The process of placing stakes in the ground at control points on a site that is being developed. For example, after you place points in your drawing, or after you design an alignment, you can create stakeout reports that list the coordinates of each stake. Someone else can then use these stakeout reports to place (or adjust) the stakes at the site.

**standard breakline**  A breakline defined from selecting consecutive points or point numbers, or selected 3D polyline or 3D line objects.

**starshot** Determining azimuth by using a star rather than a compass.

**station** A distance along a horizontal alignment.

**station equation** A point on an alignment that defines a change in the station values after that point.

**stationing** The labeling that provides a reference when talking about a specific point along the reference baseline.

**stopping sight distance** The distance required to safely stop a vehicle, traveling at design speed, to avoid a collision with any other non-moving objects obstructing the travel path.

**stratum**  The difference between two surfaces that exist in a drawing, usually the existing ground surface and a finished ground surface, which is used for calculating volumes.

**structure object** A type of object used to visually represent structural components used in a pipe network, such as a headwall or a catch basin.

**style** A logical collection of settings that applies to a class of objects. Styles simplify the process of apply settings by simply referencing a style. Modifying a style affects all the objects referencing that style.

**subassembly** An AutoCAD drawing object (AECCSubassembly) that defines the geometry of a component used in a corridor section. The AutoCAD Civil tool palette and tool catalogs provide a variety of preconfigured subassemblies, such as travel lanes, curbs, shoulders, and ditches.

**subdivision** An unimproved tract of land surveyed and divided into parcels for purposes of sales.

**sunshot** Determining azimuth by making observations on the sun rather than by a compass.

**surface** A network of elevation data (either TIN or Grid). The points of a surface are connected into either triangles or a grid, which are then used to interpolate contours, and to generate profiles and cross sections. A surface represents the ground condition at a particular time or event.

**surface border** *See* border (page 2469).

**surface boundary** *See* boundaries (page 2469).

**surface distance** The distance between two points, measured along the ground surface. On a sloped surface, the distance between two points can also be measured as a horizontal distance and a vertical distance.

**surface profile** A profile object that represents existing ground elevations along a horizontal alignment or other linear feature that supports profile views. Typically this data is derived from an existing surface or data file, and consists of a series of connected vertical tangent lines without vertical curves defined. If necessary, you can add vertical curves.

**surface smoothing** A process that interpolates and extrapolates surface data to derive additional elevation values. Kriging and Natural Neighbor Interpolation (NNI) are the two methods of surface smoothing.

**survey command language** A language that you can use either with a data collector or when you enter survey data manually. This language describes survey observations. For example, you can use the AD [VA] command to enter an angle, distance, and vertical angle.

**survey database** Contains all the control points, known directions, observation measurements, traverse definitions, figures, and standard deviations based on equipment data for the Survey project. This includes observations imported from data collector files, entered from the Survey Toolspace tab (Traverse and Observation Editors, Survey Command Window and Batch File processing).

**Survey Point** A point that is created by the Survey features of AutoCAD Civil, such as importing a field book or LandXML file containing survey data. A survey point can represent a Control Point, Non-control Point, Setup, or Sideshot. When a survey point is placed in the AutoCAD Civil drawing, it creates a special AutoCAD Civil COGO point that prevents its location and description from being modified from outside the AutoCAD Civil Survey features. For example, a survey point location may be a traverse station whose location is dependent on many observations from previous traverse stations, all originating from a known control point and direction. In the AutoCAD Civil drawing the Survey Point displays a different icon than the AutoCAD Civil COGO point to indicate its origin.

**Survey tab** The tab in Toolspace where you access and manage survey settings, survey-related databases, and survey project data.

**tangent** A straight line segment that forms part of a horizontal alignment or profile. Tangent Distances are measured as the horizontal distance between the two end points.

**target** The element of grading design criteria that determines what the grading is going to intercept. A target can be a surface, absolute elevation, relative elevation, or distance.

**target line** *See* daylight line (page 2472).

**template** A collection of default settings and styles used to create a drawing.

**terminators** Graphics, such as arrowheads, ticks, or crow's feet, that display at the end of lines, arcs, or spirals.

**tessellation** The representation of curves by using short straight line segments.

**tessellation angle** Controls the angular spacing of breaklines that are supplemented along curve segments, such as around corners. Used for surface creation from grading objects.

**tessellation spacing** Controls the spacing of supplemental breaklines that are added to a grading along straight segments. Used for surface creation from grading objects.



**tick** A component of a label that is usually a mark (or short line) inserted in a series at perpendicular angles to another object, such as a line or curve.

**TIN** Triangular Irregular Network. A TIN surface is the most common method of interpolating elevation data. The points are connected into triangles that are used to interpolate for contours, and to generate profiles and cross sections. The lines that make up the surface triangulation are called TIN lines. *See also* 3D face (page 2467).

**TIN volume surface** A differential surface created from a composite of points based on base and comparison surface. Also known as a differential surface.

**Toolspace window** A window that provides an object-oriented view of your engineering data. Toolspace is divided into four parts or tabs: Prospector tab, Settings tab, Survey tab, and Toolbox tab.

**topography** The features of the actual surface of the Earth.

**topology** A set of geometric connections between objects. Objects linked by topology maintain their relationships with one another. In AutoCAD Civil, objects that share the same topology are grouped in a site. *See also* site (page 2482).

**transit rule** A method of balancing a traverse, in which one assumes that the closing errors are caused less by the errors in the observed angles than by the errors in measured distance. Corrections are distributed according to the ratio of the latitude and departure of each leg of the traverse, to the sums of the latitude and departures of the entire traverse. Use the Transit Rule option when it is assumed that closing errors are due less to the errors in the observed angles than to errors in the measured distances.

**transition region** An area of a grading that blends the control regions on either side of it. A transition region has no design criteria assigned to it.

**transparent command** A command that you can run while another command is in progress. Transparent commands begin with an apostrophe (').

**traverse** A method of surveying in which length and directions of lines between points on the Earth are obtained by or from field measurements, and used in determining positions of the points.

Traverse closure for closed or open loops is recognized by the observation to a control (fixed or known) point. A closed traverse begins and ends at the same control point. An open traverse closes at a different control point than the beginning point.

**triangle area** The 2-dimensional (2D) area of a triangle face computed from the northing and easting of each triangle point. The total triangle area is the sum of all 2D triangle areas with the surface boundaries.

**trim (surface)** The process of removing unwanted TIN lines from a surface, thereby removing triangles.

**TS** Tangent-Spiral Intersect.

**vault** A database that is managed by Autodesk Vault.

**Vault reference** A type of data reference between drawings within a Vault project.

**VCC** Vertical Compound Curve Intersect.

**VCCE** Vertical Compound Curve Intersect Elevation.

**VCCS** Vertical Compound Curve Intersect Station.

**vertical alignment** *See* profile (page 2480).

**vertical angle** An angle above or below the horizontal plane. The angle is expressed in DDMMSS from this horizon. Vertical angles expressing a downhill slope from the instrument station are described with a negative value. Zenith angles are similar to vertical angles, except that a zenith angle has a vertical reference plane.



**vertical curve** A curve used on a profile (most commonly on layout profiles) to provide a gradual change in slope from one tangent to the other. There are three types of vertical curve: parabolic, circular, and asymmetrical.

**vertical distance** A distance measured along a sloped surface. For example, if you measure a distance from point A to point B that is on a 3:1 grade, then that distance is longer than the distance measured horizontally.



**vertical exaggeration** An increase of vertical scale relative to horizontal scale, used to make grade changes easier to differentiate. *See* vertical scale (page 2487).

**vertical scale** The scale that is compared to the horizontal scale to calculate the vertical exaggeration in profiles and cross sections. It does not actually change the scale that is used when the drawing is plotted.

**vertical to horizontal** Vertical angles and distances are converted automatically to horizontal angles and distances when you adjust a traverse loop, or perform Least Squares adjustment on observed data.

**void region** An area along a footprint where grading has not been applied, creating a gap in the grading.

**volume surface** A surface that is created by calculating volumes using the grid (differential) or TIN (composite) methods. The surface is created from the two surfaces that make up the stratum. The elevation values of a volume surface are actually the difference between the two surfaces. For example, at point 1000,1000, the bottom surface has an elevation of 100, and the top surface has an elevation of 150. The elevation of point 1000,1000 on the volume surface is the difference between the two surfaces, which is 50.

**VRC** Vertical Reverse Curve Intersect.

**VRCE** Vertical Reverse Curve Intersect Elevation.

**VRCS** Vertical Reverse Curve Intersect Station.

**wall breakline** A breakline that represents surface features such as retaining walls, curbs, bridge abutments, and so on.

**water drop** A path drawn with a 2D or 3D polyline, which represents water as it flows downhill.

**watersheds** Catchment areas for rainfall that are delineated as the drainage areas producing runoff. Base flow in a stream also usually comes from the same area.

**weeding** The removal of points along a selected polyline representing a contour. The weeding factors determine the amount of points removed. You can use weeding to reduce the amount of point information taken from the contours that may not be necessary to generate an accurate surface. *See also* weeding factors (page 2487).

**weeding factors** The settings used to reduce redundant points along the contours by ignoring contour vertices that are close together or along a straight line. A larger distance and deflection angle will weed a greater number of points.

**widening** A type of dynamic offset alignment that expands the width of a roadway for a specified length to accommodate a feature such as a turn lane or bus bay. The widening usually includes a transition region at one or both ends.

**zenith angle** The point directly overhead or the point where an observer's vertical line pierces the celestial sphere. Opposite zenith is the nadir.

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