User Guide

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1. PROGRAM DESCRIPTION

1.1. General description of the program

AutoCAD® Structural Detailing is used to prepare detailed drawings of designed steel structures. After completing the stage of architectonic structure design and generating a structure calculation model (this stage comprises calculations and verification of structure elements), the stage of preparing project technical documentation with required final drawings takes place.

AutoCAD® Structural Detailing - Steel is a program which allows generation of complete detailing documentation of steel structures (drawings + tables of materials). The process of creating drawings in **AutoCAD®** Structural Detailing - Steel is automatic. Drawings are generated based on a 3D structure model using the options available in **AutoCAD®** Structural Detailing - Steel with the possibility to apply options available in the AutoCAD® program. In case a structure model is modified, it is possible to update automatically the created documentation.

AutoCAD® Structural Detailing - Steel is a program enabling generation of detailing drawings; it is divided into a few parts which are responsible for:

- 1. link with the calculation program (*Autodesk Robot Structural Analysis*): data exchange, structure recalculation, if needed
- edition of a drawing or part of a drawing (projections, cuts, etc.) including additional drawing elements, correction of existing structure elements, adding structure element dimensions
- 3. generation of final drawings
- 4. printout management.

AutoCAD® Structural Detailing combined with **Autodesk Robot Structural Analysis** allows the user to make a complete structure project:

- generation of a structure model and structure calculations
- structure design
- generation of final drawings allowing edition of prepared drawings.

Basic tasks that are performed in **AutoCAD® Structural Detailing** - **Steel** after reading in a structure model from the calculation program, include:

- possibility to define connections of structure elements
- posibility of elements' machining (fitting to a line or object, chamfering)
- possibility to single out assemblies and groups in a structure model according to the actual procedure of structure assembly
- possibility to define user descriptions of parts
- possibility to obtain a drawing of a whole structure as well as to prepare the overall bill of materials
- · generation of detailed drawings.

The following objects have been distinguished in the program:

- Structure model, structure model elements real (3D) structure elements; for them final drawings are created
- Position an object concerned with organization of structure model elements; a (letter digit) designation of identical parts or assemblies in a structure
- Document a set of views of a position in determined projections and views (any number of documents may be generated for each position); a document consists of views. NOTE: A document may be edited only in the edition layout; a document may be printed only after it is inserted to the printout layout
- (Document) template a predefined set of views composing a logical whole (e.g. 3
 projections of a given object); each document is created based on a template
 determining which drawings (projections, sections) should be included in the document

- View a single drawing; it is always a document component if it has been added to a
 printout, it is simultaneously a printout element. NOTE: only a view (drawing) contained
 in a printout may be printed
- Printout ready-to-use printout composed of views; its equivalent in the AutoCAD®
 program is a layout together with AutoCAD views provided on it; for each printout there is
 exactly one layout corresponding to it.

The elements listed concern the stages of work on a project.

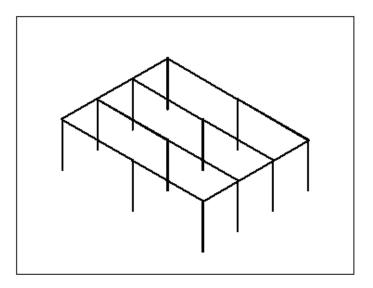
1.2. Stages of work on structure project drawings

Creating documentation of a designed structure (drawings) may be divided into the following stages:

1. Defining or loading a structure model

A structure model may be defined in AutoCAD® Structural Detailing – Steel. It is also possible to prepare it in another program and open in AutoCAD® Structural Detailing - Steel (see: AutoCAD Structural Detailing - Robot link).

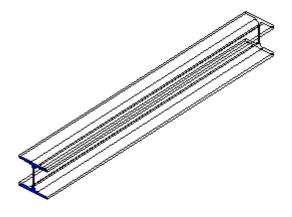
Drawings will be prepared for 3D elements of a real 3D structure.



All operations are performed in the model layout, which is an AutoCAD® object used when working with a structure model.

2. Assigning positions to structure elements

A position is an object concerned with organization of structure model elements. A position is assigned to one or several structure elements, for which a common set of drawings may be generated.

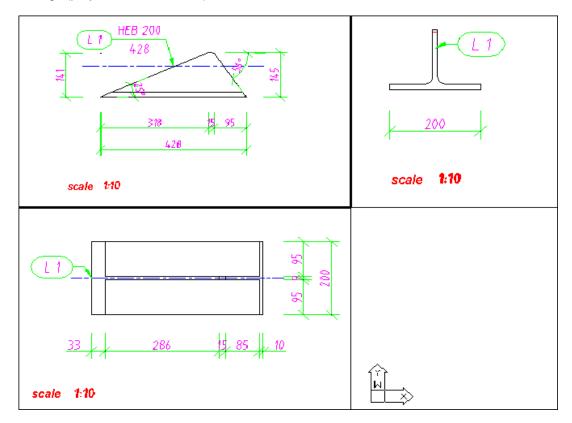


Assigning positions is available on the Model and Parts Edition tabs of the Object inspector dialog. Positions defined are displayed on the Positions tab.

3. Defining documents

A document is a set of drawings for a position (for each position, any number of documents may be generated). A document consists of views, but cannot be printed.

Documents are generated based on templates, which are predefined sets of views composing a logical whole (for example, 3 projections of a given object). A template determines which drawings (projections or sections) should be included in a document.

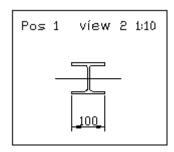


All operations are performed in edition layouts.

The edition layout is an AutoCAD® object. Only one document may be active at a time and only this document may be edited in the edition layout. You should not print the contents of the edition layout.

4. Defining (editing) views/projections

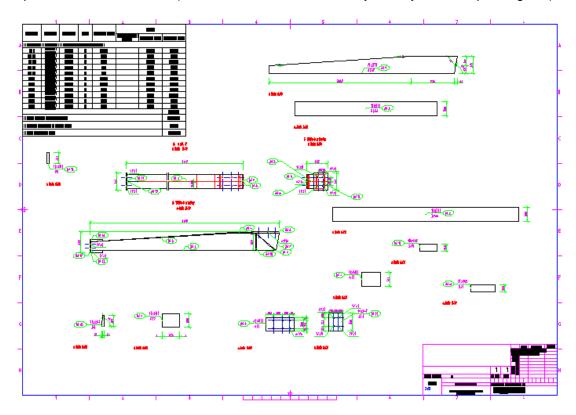
A view is a single drawing that always constitutes a document component; if it has been added to a printout, it is simultaneously a printout element. Only a view (drawing) contained in a printout may be printed.



Projections of assemblies are prepared based on the coordinate system of the main part (by default it is the longest part of an assembly). To modify projections, you can either rotate the view or, before drawings are generated, change the main part of the assembly (using the Change main part of group command on the Model tab of the Object Inspector). Edition is performed in the edition layout.

5. Generating a printout

A printout consists of views (for each Printout there is exactly one layout corresponding to it).



All operations are performed in printout layouts.

The printout layout is an AutoCAD® object. It is used for composition of a final printout. For each printout layout, there is one printout.

See also:

Example of creating documents and insertion of documents in a drawing

1.3. Options available in the menu

All menu options available in AutoCAD® Structural Detailing - Steel are displayed below. The list includes the menu path and the ribbon, toolbar icon, and command to activate each option, as well as a short description of the option.

See also: Ribbon

Workframes Create workframe

AA

Opens the Workframe dialog, where you can apply a workframe to simplify the definition of a structure model in 3D space.

Menu: Steel > Workframes > Create workframe Ribbon: ASD - Model > Elements > Workframes

Toolbar: General > Workframes Command line: RBCS_WF Add node Lets you create an additional node in the existing workframe.

Menu: Steel > Workframes > Add node

Ribbon: ASD - Model > Elements > Add node

Toolbar: General > Add node Command line: RBCS ADDNODE

Delete node Lets you delete a node in the existing workframe.

> Menu: Steel > Workframes > Delete node Ribbon: ASD - Model > Elements > Delete node

Toolbar: General > Delete node

Command line: RBCS_WF_DELNODE

Add line Lets you create an additional line in the existing workframe.

Menu: Steel > Workframes > Add line Ribbon: ASD - Model > Elements > Add line

Toolbar: General > Add line

Command line: RBCS_WF_ADDLINE

Delete line Lets you delete a line in the existing workframe.

Menu: Steel > Profile

Menu: Steel > Workframes > Delete line Ribbon: ASD - Model > Elements > Delete line

Toolbar: General > Delete line

Command line: RBCS_WF_DELLINE

Profiles Opens the Profile dialog, where you can define a bar of a selected section.

Ribbon: ASD - Model > Elements > Profile

Toolbar: General > Profile Command line: RBCS PROFILE

Opens the Plate dialog, where you can define a plate. **Plates**

Menu: Steel > Plates

Ribbon: ASD - Model > Elements > Plates

Toolbar: General > Plates Command line: RBCS PLATE

Bent profiles Opens the Bent profile dialog, where you can define a curvilinear bar of

selected profile.

Menu: Steel > Bent profiles

Ribbon: ASD - Model > Elements > Bent profiles

Toolbar: General > Bent profiles Command line: RBCS_BENT

Menu: Steel > User parts

User parts Opens the User parts dialog, where you can define (select) user objects.

Ribbon: ASD - Model > Elements > User parts

Toolbar: General > User parts Command line: RBCS_USERPART

Grates Opens the Grates dialog, where you can define a grate of specified

> dimensions and weight. Menu: Steel > Grates

Ribbon: ASD - Model > Elements > Grates

Toolbar: General > Grates Command line: RBCS_GRATE

Compound profiles

Opens the Compound profiles dialog, where you can define a compound

section bar.

Menu: Steel > Compound profiles

Ribbon: ASD - Model > Elements > Compound profiles

Toolbar: General > Compound files Command line: RBCT_MMACRO

Castellated beam

Opens the Special profiles - castellated beam dialog, where you can

define a castellated beam.

Menu: Steel > Castellated beam

Ribbon: ASD - Model > Elements > Castellated beam

Toolbar: General > Castellated beam Command line: RBCT_MMACRO

Connections

The submenu, the panel and toolbar to automatically create different types

of connections between selected elements of steel structures.

Menu: Steel > Connections

Ribbon: ASD - Model > Connections

Toolbar: All connections; Automatic connections I; Automatic connections

II; Automatic connections III; Automatic connections IV

Plate girders

The submenu, the panel and toolbar to automatically create plate girders;

using this software version you can define a column, beam and

multisegment beam.

Menu: Steel > Plate girders
Ribbon: ASD - Model > Elements
Toolbar: General > Plate girders

Machining Cut by line

Lets you cut a part (profile or plate) so that it fits the plane determined by a

line.

Menu: Steel > Machining > Cut by line

Ribbon: ASD - Model > Machining > Cut by line

Toolbar: Machining > Cut by line Command line: RBCS_CUTBYLINE

Fit to line

Lets you trim a given profile so that it fits the plane determined by a line,

perpendicular to UCS.

Menu: Steel > Machining > Fit to line

Ribbon: ASD - Model > Machining > Fit to line

Toolbar: Machining > Fit to line Command line: RBCS_FITTOLINE

Fit to polyline

Lets you trim a given part so that it fits the broken plane determined by a

polyline, perpendicular to UCS.

Menu: Steel > Machining > Fit to polyline

Ribbon: ASD - Model > Machining > Fit to polyline

Toolbar: Machining > Fit to polyline Command line: RBCS_FITTOPOLY

Fit to object

Opens the Fit to object dialog, where you can trim a given part so that it

fits another steel part.

Menu: Steel > Machining > Fit to object

Ribbon: ASD - Model > Machining > Fit to object



Toolbar: Machining Fit to object Command line: RBCS_FITOBJ

Cut by bisector



Opens the Cut by bisector dialog, where you can trim 2 intersecting profiles using a bisector of the angle between their axes so that they fit

each other.

Menu: Steel > Machining > Cut by bisector Ribbon: ASD - Model / Machining / Cut by bisector

Toolbar: Machining > Cut by bisector Command line: RBCS_CUTBISEC

Lengthen



Lets you lengthen an existing profile along its axis.

Menu: Steel > Machining > Lengthen

Ribbon: ASD - Model > Machining > Lengthen

Toolbar: Machining > Lengthen
Command line: RBCS_LENGTHEN

Shorten



Lets you shorten an existing profile along its axis.

Menu: Steel > Machining > Shorten

Ribbon: ASD - Model > Machining > Shorten

Toolbar: Machining I > Shorten
Command line: RBCS_SHORTEN

Bent plate



Lets you bend an indicated plate.

Menu: Steel > Machining > Bent plate

Ribbon: ASD - Model > Machining > Bent plate

Toolbar: Machining II > Bent plate Command line: RBCS_BENTPLATE

See also: Bending of plates

Remove bending from plate



Lets you remove an existing bend from the indicated plate. Menu: Steel > Machining > Remove bending from plate

Ribbon: ASD - Model > Machining > Remove bending from plate

Toolbar: Machining II > Remove bending from plate

Command line: RBCS_UNBENTPLATE

Split profile



Lets you divide an existing profile into several independent parts.

Menu: Steel > Machining > Split profile

Ribbon: ASD - Model > Machining > Split profile

Toolbar: Machining > Split profile Command line: RBCS_SPLIT

Merge profiles



Lets you connect independent profiles into one profile.

Menu: Steel > Machining > Merge profiles

Ribbon: ASD - Model > Machining > Merge profiles

Toolbar: Machining > Merge profiles Command line: RBCS_MERGE

Split plate



Lets you divide an existing plate into several independent parts. Specify the plate and define a cutting line (the cutting line can be defined as a line, polyline, or arc). The spacing between parts of the split plate can be defined (the spacing is divided symmetrically on both sides of the cutting line)

NOTE: If the cutting line intersects chamfers defined in the plate, these chamfers are removed from the plate. If the cutting line does not intersect

chamfers, they remain in the split parts of the plate.

Menu: Steel > Machining > Split plate

Ribbon: ASD - Model > Machining > Split plate

Toolbar: Machining > Split plate Command line: RBCS_SPLIT

Merge plates



Lets you connect 2 independent plates into 1 plate, as long as the plates

lie in one plane and touch each other.

The resulting plate (type: user) is made of the material of which the plate

selected first is made. All machinings of merged plates are kept.

Menu: Steel > Machining > Merge plates

Ribbon: ASD - Model > Machining > Merge plates

Toolbar: Machining > Merge plates Command line: RBCS_MERGE

Add plate vertex



Lets you add a vertex (or vertexes) in a polyline that defines an indicated plate. Adding a new plate vertex changes the type of the plate (to a user-defined plate).

Menu: Steel > Machining > Add plate vertex

Ribbon: ASD - Model > Machining > Add plate vertex

Toolbar: Machining > Add plate vertex Command line: RBCS_ADDVERTEX

Remove plate vertex



Lets you remove a vertex (or vertexes) belonging to a polyline that defines an indicated plate. Removing a plate vertex is possible only when the plate contour will be a closed polyline after removing the vertex (for example, it is impossible to remove a vertex from a triangular plate). Removing a plate vertex changes the type of the plate (to the user-defined

Menu: Steel > Machining > Remove plate vertex

Ribbon: ASD - Model > Machining > Remove plate vertex

Toolbar: Machingng > Remove plate vertex Command line: RBCS_REMOVEVERTEX

Chamfer plate corner



Lets you chamfer a plate corner. There are a few types of chamfering

available.

Menu: Steel > Machining > Chamfer plate corner

Ribbon: ASD - Model > Machining > Chamfer plate corner

Toolbar: Machining > Chamfer plate corner Command line: RBCS_CHAMFCORN

Insert arc



Lets you insert an arc (bent profile) between 2 indicated profiles.

After you select 2 profiles, the following question appears in the command line: 'Do you want to join profiles?' Depending on the answer to this question, the resultant profile may be:

NO: a composition of 3 profiles (2 initial profiles and a bent profile)
YES: 1 profile (3 component profiles will be joined into 1 profile)

Menu: Steel > Machining > Insert arc

Ribbon: ASD - Model > Machining > Insert arc

Toolbar: Machining > Insert arc

Command line: RBCS_BENDCORNER

Drill/Bolt



Opens the Drilling dialog, where you can define openings and bolts.

Menu: Steel > Machining > Drill/Bolt

Ribbon: ASD - Model / Bolts/Welds / Drill/Bolt

Toolbar: Machining II > Drill/Bolt Command line: RBCS_DRILL

Weld Opens the Welds dialog, where you can define welded connections.

Menu: Steel > Machining > Weld

Ribbon: ASD - Model > Bolts/Welds > Weld

Toolbar: Machining > Weld Command line: RBCS_WELD

Multi-weld

Opens the Welds dialog, where you can define welded connections for several elements simultaneously. After several objects are selected (profiles or plates), the software performs an analysis of the contact area for the indicated elements. After you specify a weld type in the Welds dialog, welded connections of the indicated elements are generated.

Menu: Steel > Machining > Multi-weld

Ribbon: ASD - Model > Bolts/Welds > Multi-weld

Toolbar: Machining II > Multi-weld Command line: RBCS_MWELD

Copy weld Lets you copy selected welds.

Menu: Steel > Machining > Copy weld

Ribbon: ASD - Model > Bolts/Welds > Copy weld

Toolbar: Machining II > Copy weld Command line: RBCS COPYWELD

Copy drills / bolts Lets you copy selected drills or bolts.

Menu: Steel > Machining > Copy drills/bolts

Ribbon: ASD - Model > Bolts/Welds > Copy drills/bolts

Toolbar: Machining II > Copy drills/bolts Command line: RBCS_COPYDRILL

Adopt drills Lets you adopt all properties of a selected drill.

Menu: Steel > Machining > Adopt drills

Ribbon: ASD - Model > Bolts/Welds > Adopt drills

Toolbar: Machining II > Adopt drills Command line: RBCS_TAKEDRILL

Insert bolts Lets you insert a bolt into a selected drill. Menu: Steel > Machining > Insert bolts

Ribbon: ASD - Model > Bolts/Welds > Insert bolts

Toolbar: Machining II > Insert bolts Command line: RBCS_BOLTHOLES

Lets you explode a group of holes/bolts into basic AutoCAD®objects. Explode group of holes/bolts

Menu: Steel > Machining > Explode group of holes/bolts

Ribbon: ASD - Model > Bolts/Welds > Explode group of holes/bolts

Toolbar: Machining II > Explode group of holes/bolts

Command line: RBCS_EXPLDRILL

Remove bolts Lets you remove bolts.

Menu: Steel > Machining > Remove bolts

Ribbon: ASD - Model > Bolts/Welds > Remove bolts

Toolbar: Machining > Remove bolts Command line: RBCS_REMBOLTS

Copy connection Lets you copy a selected connection.

Menu: Steel > Machining > Copy connection

Ribbon: ASD - Model > Connections > Copy connection

Toolbar: Machining II > Copy connection Command line: RBCS_CONNECTION

See also: Copy connection







Update connections



Lets you update selected connections after modifications in structure geometry. Connections are adjusted to new structure dimensions; the connection update mode depends on the option settings in the Project

preferences dialog.

Menu: Steel > Machining > Update connections

Ribbon: ASD - Model > Connections > Update connections

Toolbar: Machining II > Update connections Command line: RBCS CONNUPDATE See also: Project preferences / General

Tools **Explode**

Lets you explode some complex elements (groups, connections) into

single elements.

Menu: Steel > Tools > Explode

Ribbon: ASD - Model > Tools > Explode

Toolbar: General > Explode Command line: RBCS_EXPLODE

Modify properties

Lets you modify parameters of selected objects; a dialog displays, where

you can change parameter values for a selected object.

Menu: Steel > Tools > Modify properties

Ribbon: ASD - Model > Tools > Modify properties

Toolbar: General > Modify properties Command line: RBCS PROPERTIES

Family Manager

Opens the Family Manager dialog; use this option to manage families of

steel elements.

Menu: Steel > Tools > Family Manager

Ribbon: ASD - Model > Tools > Family Manager

Toolbar: General > Family Manager Command line: RBCS_FAMILIES

Group

Lets you create groups of selected objects.

Menu: Steel > Tools > Group

Ribbon: ASD - Model > Groups > Group

Toolbar: General > Group Command line: RBCS_GROUP

Group assemblies

Lets you group assemblies based on the defined connections.

Menu: Steel > Tools > Group assemblies

Ribbon: ASD - Model > Groups > Group assemblies

Toolbar: General > Group assemblies Command line: RBCS ASSEMBLY

User sections

Lets you define a profile and save it to a profile database or locally, in a

project.

See also: Example of defining a user section

Menu: Steel > Tools > User sections

Ribbon: ASD - Model / Tools / User sections

Toolbar: General > User sections Command line: RBCS USRPROFILE See also: Example of user profile definition

Inquire Object info

Opens the ACAD text window, where you can get information about the

selected object.

Menu: Steel > Tools > Inquire > Object info Ribbon: ASD - Model > Tools > Object info

Toolbar: General > Object info

Command line: RBCS_INQ_ELEMENT



Group

H

Opens the ACAD text window, where you can get information about the

group to which the selected object belongs. Menu: Steel > Tools > Inquire > Group Ribbon: ASD - Model > Tools > Group

Toolbar: General > Group

Command line: RBCS_INQ_GROUP

Assembly



Opens the ACAD text window, where you can get information about the

assembly to which the selected object belongs. Menu: Steel > Tools > Inquire > Assembly Ribbon: ASD - Model > Tools > Assembly

Toolbar: General > Assembly

Command line: RBCS_INQ_ASSEMBLY

Center of gravity



Opens the ACAD text window, where you can calculate coordinates of the

gravity center for a structure or its selected part.

Menu: Steel > Tools > Inquire > Center of gravity

Ribbon: ASD - Model > Tools > Center of gravity

Toolbar: General > Center of gravity Command line: RBCS_INQ_COG

Model size



Opens the ACAD text window, where you can calculate dimensions and

weight of a structure or its selected part.

Menu: Steel > Tools > Inquire > Model size
Ribbon: ASD - Model / Tools / Model size
Command line: RBCS INQ MSIZE

Collision detection



Lets you find a collision of structure components (profiles, plates, bolts,

and so on).

The command line displays a number of detected collisions between

structure elements. It is possible to save collision tests in the

RBCS_Collision_Detection layer and represented by AutoCAD® 3D Solid

obiects.

Menu: Steel > Tools > Inquire > Collision detection Ribbon: ASD – Model / Tools / Collision detection

Command line: RBCS_INQ_COLISION

See also: Collision detection

Filters Show current



Lets you filter objects in a drawing (only selected objects are displayed).

The filter is also accessible from the Object Inspector dialog.

Menu: Steel > Tools > Filters > Show current selection

Ribbon: ASD - Model > Tools > Show current selection

Toolbar: Workspace > Show current selection Command line: RBCT_SHOWCURSEL

Hide current selection



Lets you filter objects in a drawing (only unselected objects are displayed).

The filter is also accessible from the Object Inspector dialog. Menu: Steel > Tools > Filters > Hide current selection Ribbon: ASD - Model > Tools > Hide current selection

Toolbar: Workspace > Hide current selection Command line: RBCT_HIDECURSEL

Show all



Lets you filter objects in a drawing (all objects are displayed). The filter is

also accessible from the Object Inspector dialog.

Menu: Steel > Tools > Filters > Show all Ribbon: ASD - Model > Tools > Show all

Toolbar: Workspace > Show all

Command line: RBCT_SHOWALLOBJ

Workspace Show / Hide Inspector



Lets you show or hide the Object Inspector dialog which is by default

displayed in the left part of the screen.

Menu: Steel > Tools > Workspace > Show / Hide Inspector Ribbon: ASD - Model > Tools > Object inspector - Show / Hide

Toolbar: Workspace > Object inspector - Show / Hide

Ribbon: ASD - Model > Tools > Object snap settings

Command line: RBCTOI

Object snap settings



Opens the **Snap Mode** dialog; use this option to set active snap points for steel profiles.

Menu: Steel > Tools > Workspace > Object snap settings

Toolbar: Workspace > Object snap settings

Command line: RBCS_PSNAP

Lock X direction



Lets you select or deselect blocking the local X axis of the bar profile; if this option is selected, you can perform the operation of extending a bar only along the bar axis; if this option is deselected, a profile can be

extended in any direction you selected.

Menu: Steel > Tools > Workspace > Lock X direction Ribbon: ASD - Model > Tools > Lock X direction

Toolbar: Workspace > Lock X direction

Command line: RBCS_LOCKX

2D copying / moving

Options for moving or copying selected elements in a specified direction

(along the axis or in the plane).

Menu: Steel > Tools > Workspace > 2D copying/moving

Ribbon: ASD - Model > Tools

Toolbar: Workspace > 2D copying/moving

Select assembly



Lets you select a part of an assembly; it selects a whole assembly that this

part belongs to.

Menu: Steel > Tools > Workspace > Select assembly Ribbon: ASD - Model > Tools > Select assembly

Toolbar: Workspace > Select assembly Command line: RBCS_SELASSEMBLY

Dvnamic UCS



Lets you fit the view plane and active UCS to the UCS of a selected

object.

Menu: Steel > Tools > Workspace > Dynamic UCS Ribbon: ASD - Model > Tools > Dynamic UCS

Toolbar: Workspace > Dynamic UCS Command line: RBCS_DYNUCS

Elevation mark - model



Lets you insert an elevation mark in a selected plane and in a selected

location in the drawing

Menu: Steel > Tools > Workspace > Elevation mark - model Ribbon: ASD - Model > Tools > Elevation mark - model

Toolbar: Workspace > Elevation mark - model

Command line: RBCS_MELEV

Clipping plane on Lets you limit the model view with 2 planes

Menu: Steel > Tools > Workspace > Clipping plane on Ribbon: ASD - Model > Tools > Clipping plane on

Toolbar: Workspace > Clipping plane on

Command line: RBCS_CLIPON

Clipping plane off Lets you deselect limiting the model view with 2 planes.

Menu: Steel > Tools > Workspace > Clipping plane off Ribbon: ASD - Model > Tools > Clipping plane off

Toolbar: Workspace > Clipping plane off

Command line: RBCS_CLIPOFF

Positions

Assign position Opens the Manual positioning dialog, where you can ascribe the position

to a selected part.

Menu: Steel > Positions > Assign position

Ribbon: ASD - Model > Positions > Assign position

Toolbar: General > Assign position Command line: RBCS_MANUALPOS

Automatic positioning

structures

Opens the Automatic positioning dialog, where you can ascribe positions

to structure elements automatically.

Menu: Steel > Positions > Automatic positioning

Ribbon: ASD - Model > Positions > Automatic positioning

Toolbar: General > Automatic positioning Command line: RBCS_AUTOPOS

Remove position Lets you remove the ascribed position from a selected part.

Menu: Steel > Positions > Remove position

Ribbon: ASD - Model > Positions > Remove position

Toolbar: General > Remove position Command line: RBCS_REMPOS

Verify positions Opens the Position verification dialog, where you can run verification of

selected positions.

Menu: Steel > Positions > Verify positions

Ribbon: ASD - Model > Positions > Verify positions

Toolbar: General > Verify positions Command line: RBCS_VERIFYPOS

Parametric A submenu and toolbar with options that enable definition of typical

elements of steel structures.

Menu: Steel > Parametric structures

Ribbon: ASD – Model / Parametric structures

Toolbar: Parametric structures

Multi-bay frame Lets you define a parametric multi-bay frame.

Menu: Steel > Parametric structures > Multi-bay frame

Ribbon: ASD - Model > Parametric structures > Multi-bay frame

Toolbar: Parametric structures > Multi-bay frame

Roof truss Lets you define a parametric roof truss.

Menu: Steel > Parametric structures > Roof truss

Ribbon: ASD - Model > Parametric structures > Roof truss

Toolbar: Parametric structures > Roof truss

Bracing Lets you define a parametric bracing.

Menu: Steel > Parametric structures > Bracing

Ribbon: ASD - Model > Parametric structures > Bracing

Toolbar: Parametric structures > Bracing

Brace Lets you define a parametric brace.

Menu: Steel > Parametric structures > Brace

Ribbon: ASD - Model > Parametric structures > Brace

Toolbar: Parametric structures / Brace

Lets you define parametric stairs. **Stairs**

Menu: Steel > Parametric structures > Stairs

Ribbon: ASD - Model > Parametric structures > Stairs

Toolbar: Parametric structures > Stairs

Stairs spiral Lets you define parametric spiral stairs.

Menu: Steel > Parametric structures > Stairs spiral

Ribbon: ASD - Model > Parametric structures > Stairs spiral

Toolbar: Parametric structures > Stairs spiral

Staircase Lets you define a parametric staircase.

Menu: Steel > Parametric structures > Staircase

Ribbon: ASD - Model > Parametric structures > Staircase

Toolbar: Parametric structures > Staircase

Railing Lets you define a parametric railing.

Menu: Steel > Parametric structures > Railing 4111

Ribbon: ASD - Model > Parametric structures > Railing

Toolbar: Parametric structures > Railing

Ladder Lets you define a parametric ladder.

Menu: Steel > Parametric structures > Ladder

Ribbon: ASD - Model > Parametric structures > Ladder

Toolbar: Parametric structures > Ladder

Cage Lets you define a parametric cage.

Menu: Steel > Parametric structures > Cage

Ribbon: ASD - Model > Parametric structures > Cage

Toolbar: Parametric structures > Cage

Purlin spreading Lets you define a distribution of purlins.

Menu: Steel > Parametric structures > Purlin spreading

Ribbon: ASD - Model > Parametric structures > Purlin spreading

Toolbar: Parametric structures / Purlin spreading

Grate spreading Lets you define a distribution of grates.

Menu: Steel > Parametric structures > Grate spreading

Ribbon: ASD - Model > Parametric structures > Grate spreading

Toolbar: Parametric structures > Grate spreading

Drawings edition

Change style



Opens the Design styles dialog, where you can change a design style for

a selected drawing.

Menu: Steel > Edit drawings > Change style

Ribbon: ASD - Drawings > Edit drawings > Change style

Toolbar: Edit drawings > Change style Command line: RBCS CNGSTYLE

Adjust style



Opens the Design styles dialog, where you can change a design style for

a selected drawing.

Menu: Steel > Edit drawings > Adjust style

Ribbon: ASD - Drawings > Edit drawings > Adjust style

Toolbar: Edit drawings > Adjust style Command line: RBCS_ADJSTYLE

Lock dimensions



Lets you freeze generated drawings with existing dimensions or descriptions. This is the option to protect existing drawings (changed manually by the user) from losing the changes while the drawing is updated. Automatically generated drawings are quite often changed by adding, deleting or moving dimensions or descriptions. To avoid losing changes while the drawing is updated, select this option and lock dimensions in the drawing.

After executing this command, the command line displays a message about dimensions in the drawing (locked dimensions and dimensions that are not locked); in addition, an icon displays in the Object Inspector pane.

Menu: Steel > Drawings Edition > Lock dimensions

Ribbon: ASD - Drawings > Edit drawings > Lock dimensions

Toolbar: Edit drawings > Lock dimensions

Command line: RBCS_DIMLOCK

Edit view in full screen



Select this in order to edit a created drawing on the full monitor screen.

Menu: Steel > Drawings Edition > Edit view in full screen

Ribbon: ASD - Drawings > Edit drawings > Edit view in full screen

Toolbar: Edit drawings > Edit view in full screen Command line: RBCT_FULLSCREEN_ON

Full screen off



Select this to turn off the ability to edit a drawing on the full monitor screen.

Menu: Steel > Drawings Edition > Full screen off

Ribbon: ASD - Drawings > Edit drawings > Full screen off

Toolbar: Edit drawings > Full screen off Command line: RBCT_FULLSCREEN_OFF

Insert weld symbol



Ribbon: ASD - Drawings > Edit drawings > Insert weld symbol

Toolbar: Edit drawings > Insert weld symbol

Command line: RBCS_ADDWELD

Insert elevation



Lets you insert an elevation mark in a drawing.

Menu: Steel > Drawings Edition > Insert elevation mark

Ribbon: ASD - Drawings > Edit drawings > Insert elevation mark

Toolbar: Edit drawings > Insert elevation mark

Command line: RBCS_ADDELEV

Insert part label

Lets you insert description of parts in a drawing. Menu: Steel > Drawings Edition > Insert part label

Ribbon: ASD - Drawings > Edit drawings > Insert part label

Toolbar: Edit drawings > Insert part label Command line: RBCS_PARTLBL

Insert assembly

label

Lets you insert an assembly description in a drawing.

Menu: Steel > Drawings Edition > Insert assembly label

Pibbon: ASD Drawings > Edit drawings > Insert assembly

Ribbon: ASD - Drawings > Edit drawings > Insert assembly label

Toolbar: Edit drawings > Insert assembly label

Command line: RBCS_ASLBL

Add view (detail)

Lets you create a new view (view of a section detail) in a drawing.

Menu: Steel > Drawings Edition > Add view (detail)

Ribbon: ASD - Drawings > Edit drawings > Add view (detail)

Toolbar: Edit drawings > Add view (detail) Command line: RBCS_ADDDETAILVIEW

Add cut

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Lets you create a new view (view of a section) in a drawing. To create a

cut:

1. specify points defining the cut

2. specify a range3. specify a cut name

4. select a drawing template.

Menu: Steel > Drawings Edition > Add cut

Ribbon: ASD - Drawings > Edit drawings > Add cut

Toolbar: Edit drawings > Add cut Command line: RBCS_ADDCUT

Automatic cuts

Lets you automatically create new views (cuts) in the drawing.

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Menu: Steel > Drawings Edition > Automatic cuts

Ribbon: ASD - Drawings > Edit drawings > Automatic cuts

Toolbar: Edit drawings > Automatic cuts Command line: RBCS_AUTOCUT

Move cut

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Lets you change a cut in the drawing. You can change a position or range

of the cut, and a symbol description.

Menu: Steel > Drawings Edition > Move cut

Ribbon: ASD - Drawings > Edit drawings > Move cut

Toolbar: Edit drawings > Move cut Command line: RBCS MOVCUT

Change view scale

1:10 1:20 Lets you change scale of a view (projection) in a drawing. Menu: Steel > Drawings Edition > Change view scale

Ribbon: ASD - Drawings > Edit drawings > Change view scale

Toolbar: Edit drawings > Change view scale

Command line: RBCS_CNGSCALE

Rotate view

Lets you rotate a view (projection) in a drawing. Menu: Steel > Drawings Edition > Rotate view

Ribbon: ASD - Drawings > Edit drawings > Rotate view

Toolbar: Edit drawings > Rotate view

Command line: RBCS_ROTATEVIEWPORT

Shorten drawing



Lets you shorten a selected element in a drawing. The option is useful for long elements (in particular, for these element segments that do not contain any snap points); the element shortening is displayed as a gap in

the element subjected to shortening.

Menu: Steel > Drawings Edition > Shorten drawing

Ribbon: ASD - Drawings > Edit drawings > Shorten drawing

Toolbar: Edit drawings > Shorten drawing Command line: RBCS SHORTDRAW

Delete all shortenings



Lets you delete all the defined element shortenings in a drawing.

Menu: Steel > Edit drawings > Delete all shortenings

Ribbon: ASD - Drawings > Edit drawings > Delete all shortenings

Toolbar: Edit drawings > Delete all shortenings Command line: RBCS_DELSHORTDRAW

Dimensioning General / Detailed

Lets you refresh the view (including the dimensioning of elements) based on the settings of styles that are associated with details. You can select the following dimensioning style: standard, detailed or schematic.

Menu: Steel > Drawings Edition > Dimensioning General > Detailed Ribbon: ASD - Drawings > Edit drawings > Dimensioning General >

Detailed

Toolbar: Edit drawings > Dimensioning General > Detailed

Command line: RBCS_CNGDET

Explode drawing



Lets you explode drawings into basic AutoCAD®objects. Menu: Steel > Drawings Edition > Explode drawing

Ribbon: ASD - Drawings > Edit drawings > Explode drawing

Toolbar: Edit drawings > Explode drawing

Command line: RBCS_EXPLODE

Drawing template designer



Lets you create and modify drawing templates.

Menu: Steel > Drawings Edition > Drawing template designer Ribbon: ASD - Drawings > Generate drawings > Drawing template

designer

Toolbar: Edit drawings > Drawing template designer

Command line: RBCS_ADDTEMPLATE

Automatic printouts



Lets you create automatic printouts (one part in one drawing). NOTE: This

option is not available for schemes of elements disposition.

Menu: Steel > Automatic printouts

Ribbon: ASD - Drawings > Generate drawings > Automatic printouts

Toolbar: Printout > Automatic printouts Command line: RBCS_AUTOPRINT

Automatic printing



Lets you generate workshop drawings automatically to plotter.

Ribbon: ASD - Drawings > Generate drawings > Automatic printing

Toolbar: Printout > Automatic printout

Command line: _AUTOPLOT

Engineering drawings Engineering drawings



Lets you define a group of structure elements to make a scheme drawing of a structure model. Drawings presenting structure elements in a

simplified way are created for a defined scheme.

Menu: Steel > Engineering drawings > Engineering drawings Ribbon: ASD - Model > Groups > Engineering drawings Toolbar: Engineering drawings > Engineering drawings

Command line: RBCS_SCHEME

Create mark



Lets you create a mark, that is an object consisting of selected structure elements. It is usually an element (composed of parts such as plates, profiles, user parts or connectors) which has a different function in structure calculations, such as column, beam, frame or bracing.

Marks that have the same function in the structure and that have the same geometry can be labeled with the same name (referred to as a mark). The degree of similarity of elements (and of names of marks) is user-defined.

The basic functionality of a mark is that you can draw and describe it as a single element in the drawing.

To define a mark:

- 1. select elements that should belong to the mark
- 2. specify a mark name
- 3. define values of forces.

Menu: Steel > Engineering drawings > Create mark Ribbon: ASD - Model > Groups > Create mark Toolbar: Engineering drawings > Create mark

Command line: RBCS MARK

Create mark from assembly



Lets you create marks form all assemblies in the selection. If single elements are also selected, they are ignored while creating a mark.

To create a mark:

- 1. select assemblies
- 2. specify a mark name
- define values of forces.

Menu: Steel > Engineering drawings > Create mark from assembly Ribbon: ASD - Model > Groups > Create mark from assembly Toolbar: Engineering drawings > Create mark from assembly

Command line: RBCS_ASMARK

Create mark from single part



Lets you create marks with the same names, that are entirely included in the selection of elements. Marks created using this option consist of a single element set.

To create a mark:

- 1. select parts
- 2. specify a mark name
- 3. define values of forces.

Menu: Steel > Engineering drawings > Create mark from single part Ribbon: ASD - Model > Groups > Create mark from single part Toolbar: Engineering drawings > Create mark from single part

Command line: RBCS_SPMARK

Tube development

Lets you create a drawing of a developed tube profile.

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Menu: Steel > Tube development

Ribbon: ASD - Drawings > Printout > Tube development

Toolbar: Printout > Tube development

Styles Dimensioning style

Opens the Dimensioning styles dialog, where you can determining the styles of dimensioning for elements provided in a drawing.

Menu: Steel > Styles > Dimensioning styles

Ribbon: ASD - Drawings Styles > Dimensioning styles

Command line: RBCS_DIM

Description styles Opens the Styles of descriptions dialog, where you can determine styles

of descriptions for elements provided in a drawing.

Menu: Steel > Styles > Description styles

Ribbon: ASD - Drawings > Styles > Dimensioning styles

Command line: RBCS_DESCRIPTION

Tables

Bill of materials Lets you prepare tables of all structure elements.

Menu: Steel > Tables > Bill of materials

Ribbon: ASD - Drawings > Tables > Bill of materials

Toolbar: General > Bill of materials

Command line: RBCS_BILL

List of profiles Lets you prepare tables of profiles.

Menu: Steel > Tables > List of profiles

Ribbon: ASD - Drawings > Tables > List of profiles

Toolbar: General > List of profiles Command line: RBCS_PROFILELIST

List of profiles (by section types)

Lets you create tables of profiles by profile cross-sections.

Menu: Steel > Tables > List of profiles by section types

Ribbon: ASD - Drawings > Tables > List of profiles by section types

Toolbar: General > List of profiles by section types

Command line: RBCS_PROFILELISTSUM

Lets you prepare tables of plates.

Menu: Steel > Tables > List of plates

Ribbon: ASD - Drawings > Tables > List of plates

Toolbar: General > List of plates Command line: RBCS_PLATELIST

List of user parts Lets you prepare tables of user parts.

Menu: Steel > Tables > List of user parts

Ribbon: ASD - Drawings > Tables > List of user parts

Toolbar: General > List of user parts
Command line: _RBCS_USEROBJLIST

List of assemblies Lets you prepare assembly tables.

Menu: Steel > Tables > List of assemblies

Ribbon: ASD - Drawings > Tables > List of assemblies

Toolbar: General > List of assemblies

Command line: RBCS_ALIST

Mounting list Lets you prepare mounting tables.

Menu: Steel > Tables > Mounting list

Ribbon: ASD - Drawings > Tables > Mounting list

Toolbar: General > Mounting list Command line: RBCS_MONTLIST

Bolt list Lets you prepare connector tables.

Menu: Steel > Tables > Bolt list

Ribbon: ASD - Drawings > Tables > Bolt list

Toolbar: General > Bolt list Command line: RBCS_BLIST

Update table

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Lets you update a table presented in a drawing.

Menu: Steel > Tables > Update table

Ribbon: ASD - Drawings > Tables > Update table

Toolbar: General > Update table Command line: RBCS_ACTTABLE

Tables manager



Opens the Table printout manager dialog, where you can define/modify the appearance of the printout of tables used to prepare steel summary

tables.

Menu: Steel > Tables > Tables manager

Ribbon: ASD - Drawings > Tables > Tables manager

Toolbar: General > Tables manager Command line: RBCS_LISTPRINT

Styles Opens the Steel summary tables - style manager dialog, where you can

define/modify tables used to prepare steel summary tables.

Menu: Steel > Tables > Styles

Ribbon: ASD - Drawings > Settings > Table styles

Command line: RBCS_LISTTEMPL

Static analysis -Autodesk Robot Structural Analysis Lets you export a structure model to the calculation program Autodesk

Robot Structural Analysis.

Menu: Steel > Static analysis - Robot

Ribbon: ASD - Model > Tools > Static analysis - ARSA

Toolbar: General > Static analysis - ARSA

Command line: RBCS_R2R

DSTV data export



Lets you read / save a DSTV format file. Menu: Steel > DSTV data export

Ribbon: ASD - Model > Tools > DSTV data export

Toolbar: General > DSTV data export

Command line: RBCS_DSTV

Save model as ACIS solids



Lets you save a structure model to a DWG format file (ACIS format). You can select the full (complete description of solid details) or simplified save option. If the simplified save option is selected, profiles are saved without fillets and openings (except anchor openings), bolts, washers, nuts and

anchors are ignored.

Menu: Steel > Save model as ACIS solids

Ribbon: ASD - Model > Tools > Save model as ACIS solids

Toolbar: General > Save model as ACIS solids Command line: RBCS_EXPORTMODEL

Preferences



Opens the Options dialog, where you can specify parameters of the work

environment for AutoCAD® Structural Detailing.

Menu: Steel > Preferences

Ribbon: ASD - Model > Settings > Preferences

Toolbar: General > Preferences Command line: RBCS_PREF

Project preferences



Opens the Project preferences dialog, where you can adopt basic parameters of the current project in AutoCAD® Structural Detailing.

Menu: Steel > Project preferences

Ribbon: ASD - Model > Settings > Project preferences

Toolbar: General > Project preferences Command line: RBCS_PRJPREF

Divide project

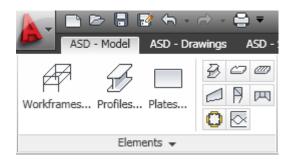


Lets you divide a project into several parts (distributed work on a document); it allows decreasing the size of a file with project data.

Menu: Steel > Divide project Toolbar: General > Divide project Command line: RBCT_DISTRIBUTE

1.4. Ribbon

The ribbon is an element of the user interface which replaces the traditional menu and toolbars and allows easy managing and adjusting the workspace.



The ribbon consists of several panels, grouped on tabs that are named by task or subject. The ribbon panels include many commands that have been on toolbars and in dialogs so far, such as icons, drop-down lists, sliders, text fields and other elements characteristic of a given tab. Using the ribbon, you do not have to display many toolbars; thus the application displays fewer functions and increases the allowable workspace placing the whole interface on a small area that can be anytime shown or hidden.

The ribbon displays automatically when a drawing is created or opened using the ASD steel workspace. You can display the ribbon manually using either of the following methods:

- select the main menu Tools > Palettes > Ribbon
- type RIBBON in the command line to show the ribbon or RIBBONCLOSE to hide it.

You can customize the ribbon, that is you can add, delete and modify positions of panel elements, in the Customize User Interface (CUI) editor window. Open this editor using either of the following methods:

- click on the Manage tab > Customization > User Interface
- type CUI in the command line.



NOTE:

You can display the ribbon horizontally, vertically or as a floating palette.

Using the editor you can also switch between workspaces (such as the classic workspace without the ribbon). To do it, select the Customize tab > Workspaces and select Set current from the context menu.

To change between workspaces, you can also use the Workspace Switching icon at the bottom right corner of the screen.

1.5. Divide project (distributed work)

Use this option to divide a file with a structure model into smaller files.

Access the option from:

- Menu: Steel / Divide project
- Toolbar: General Divide project
 Command line: RBCT DISTRIBUTE
- When running automatic drawings, select Start distributed work.

The option creates a subfolder where, in separate DWG files, information about positions is saved. The main DWG file is therefore smaller, and loading such a file takes up much less time than loading one file that contains information about all structure elements.



NOTE:

If you have already divided a DWG file (has started distributed work on the project), there is no possibility to return to one large DWG file that contains information about all structure elements.

To start distributed work:

- 1. Open a model that has defined positions.
- 2. Click the Divide project option
- 3. Select positions to be saved to external files.
- 4. Press Enter.

After a project is divided, successive positions / drawings added to a project are saved in separate files (Divide project works for new positions / drawings).



NOTE:

To copy a created project in which distributed work has begun, copy the subfolder with separate DWG files that contain descriptions of individual positions / drawings.

2. CONFIGURATION

2.1. Project preferences

2.1.1. Project preferences

Use this option to adopt basic parameters applied in AutoCAD® Structural Detailing (these parameters are saved in a DWG file).

To begin working with project preferences, open the Project preferences dialog from:

- Menu: Steel / Project preferences
- Ribbon: ASD Model / Settings / Project preferences
- Toolbar: General / Project preferences
- Command line: RBCS_PRJPREF.

To specify project preferences:

 In the left panel, use the selection tree (shown below) to specify a project preference option



In the right panel, specify parameters for the option selected on the left.

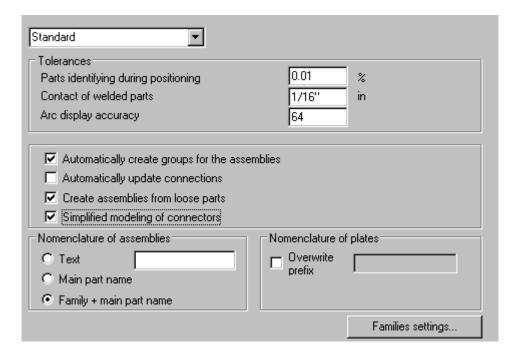
Use the buttons on the right of the dialog:

- Default click this to save values of the project preference parameters as default values.
- Save if you click this, the current project preference parameters are saved under the name specified in the General dialog.
- Delete if you click this, you delete a set of project preferences saved under the name currently displayed in the General dialog.

See also: Preferences

2.1.2. **General**

After you click General from the left pane of the Project Preferences dialog, the parameter options are as shown.

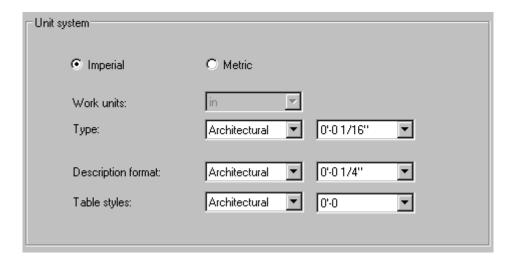


Define parameters:

- Under Tolerances:
 - Parts identifying during positioning an admissible deviation in dimensions of 2 parts which are to be treated as identical during positioning
 - Contact of welded parts an admissible distance between edges of the parts during detection of a contact line (used while welding the parts)
 - Arc display accuracy a number of straight segments (with respect to a round angle - 360 degrees) into which a created arc is divided
- Automatically create groups for the assemblies if this is selected, the elements joined by means of workshop connectors (bolts or welds) are grouped automatically while these connectors are being inserted
- Automatically update connections if this is selected, then after the structure geometry is modified, the existing connections are adjusted to new structure dimensions. Selecting the option is recommended for small structures (and fast computers); for a large structure model, the operation of updating the structure may be very time-consuming
- Create assemblies from loose parts if this is selected, all parts that do not belong to any assembly will be ascribed a new position type called Assembly (during automatic generation of assemblies). If this is not selected, the position type of parts not belonging to any assembly will not be changed
- Simplified modeling of connectors if this is selected, all connectors are displayed in a simplified way; it decreases computer memory use, particularly for large structures.
- Under Nomenclature of assemblies, you can select names of assemblies created automatically in the software
 - Text an assembly name will consist of any user-defined text
 - Main part name an assembly name will consist of a name of main element (for example, HP14x102)
 - Family + main part name an assembly name will consist of a family name and a main element name (for example, Beam_HP14x102)
- Nomenclature of plates
 - Overwrite prefix if this is selected, a name defined in the field overwrites the default prefix for plates (the name created automatically when defining a plate)
- Families settings click this to open the Family manager dialog.

2.1.3. Units

After selecting the *Units* option from the selection tree in the Project Preferences dialog, the following options display in the right part of this dialog.



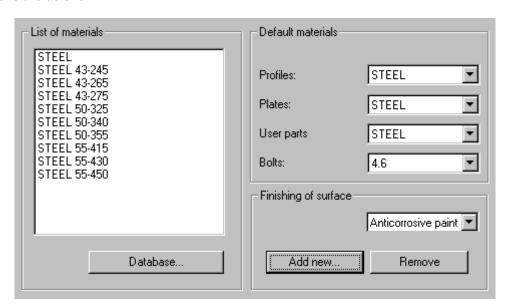
Use this tab to define **AutoCAD® Structural Detailing** work units. You can select one of the following unit systems:

- imperial
 - type:
 - o architectural (0'-0)
 - o engineering (0'-0")
- metric.

The unit system settings can be selected for the description format and table styles. You can also change the displayed number format. Use the selection lists to select a number of decimal places for each quantity. To change a number of decimal places, select an appropriate symbol from the selection list.

2.1.4. Materials

After you click Materials from the left pane of the Project Preferences dialog, the parameter options are as shown.



Specify parameters:

- Under List of materials is a list of materials available in the current project. The same material list is included on the lists accessible under Default materials (profiles, plates, user parts).
- Click Database to open the Material database dialog, where you can select materials to add to the list.
- Under Default materials, you can select a material from lists of profiles, plates, user parts, and bolts. The selected materials will be adopted by default in the dialogs for definition of profiles, plates, user parts, or bolts.



A default bolt grade chosen in the above dialog is taken into account in the AutoCAD® Structural Detailing macros (connections, typical structures) when bolts are inserted automatically. If you insert bolts 'manually', the most recently selected bolt grade is remembered in the dialog.

Under Finishing of surface, you can define and/or select types of finishing of steel element surfaces (hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat, and none). The surface finishing type is an additional attribute of structure objects (profiles, plates, assemblies).

This section includes the following buttons:

- Add new opens the Add new dialog for defining a name of a new type of surface finishing
- Remove removes a selected surface finishing type from the selection list



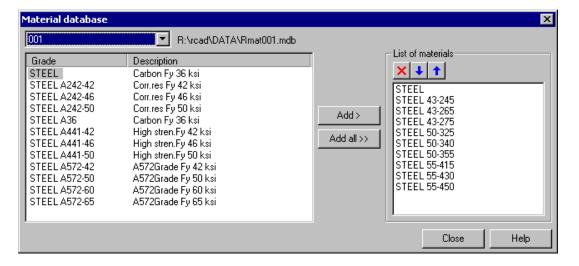
A type of surface finishing is saved in the current project, in a list of types that is saved in a DWG file.



An assembly adopts a surface finishing type from the type selected for the main part of that assembly.

2.1.5. Material database

The Material database dialog displays after you click Database on the Materials tab of the Project preferences dialog.



In the dialog, you can add materials from databases accessible in the program to the list of available materials. To add a material to the list:

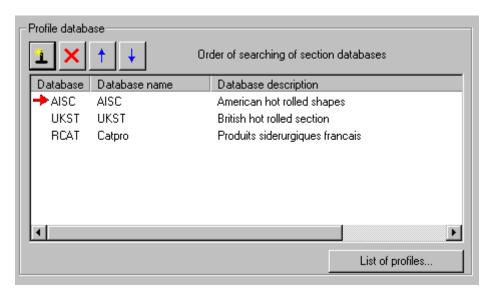
- Choose a material database from the selection list provided in the top part of the dialog (for example, 001, an American material database).
- Select a material type in the left pane
- Click Add >.

Clicking Add all >> adds all the materials included in the left pane.

Clicking deletes a selected material from the list of available materials.

2.1.6. Profiles

After you click Profiles from the left pane of the Project Preferences dialog, the parameter options are as shown.



The Profile database list displays profile folders (databases) available in the current project. Click List of profiles to open the dialog in which profiles applied in a given project are displayed.

The following icons are at the top of the dialog:

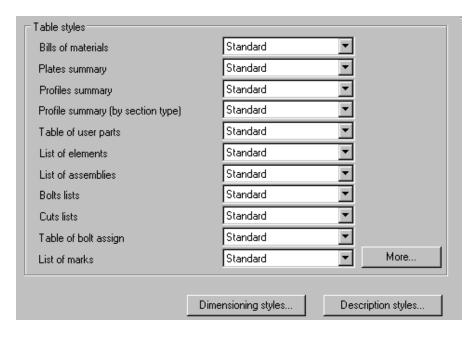
- click this to open the dialog where you can select a database to be added to the list of available profile databases

_ click this to delete a selected profile database

— click these to move a database one position up or down on the list. The order is significant while searching through available profile databases.

2.1.7. Styles

After you click Syles from the left pane of the Project Preferences dialog, the parameter options are as shown.



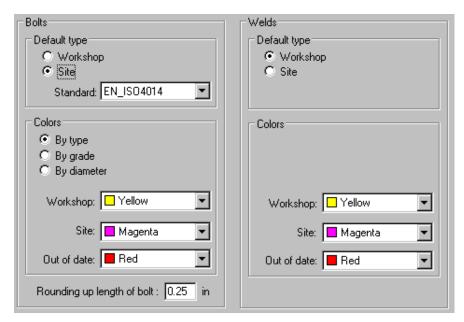
Use these parameters to define table styles for the table types available in the software (Bills of materials, Profile summary, Plate summary, and so on). The selection lists include all the styles defined in a project.

Click More... to open the Steel summary tables – style manager dialog.

At the bottom of the dialog are 2 buttons: Dimensioning styles and Description styles, which, when clicked, will open the dialogs Dimensioning styles and Styles of descriptions, respectively.

2.1.8. Connectors

After you click Connectors from the left pane of the Project Preferences dialog, the parameter options are as shown.



Define parameters of bolts and welds:

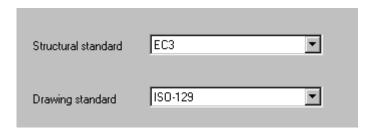
- For Default type, select a default weld/bolt type (workshop or site)
- Under Bolts, you can select a standard (database) of bolts used in the software
- Under Colors, you can specify the color of the following types of connectors displayed in drawings: workshop, site, or out-of-date (these are no longer current connectors after changes have been made in a structure model)

Specify bolt parameters:

- Select a method of assigning colors to bolts: by type, by grade, or by diameter.
- Determine a rounding up length of bolts in the software, the bolt length is automatically calculated for connections of elements. The default value for rounding up length is 5 mm, which you can modify. A value of this parameter is saved with the project in a DWG file.

2.1.9. Standards

After you click Standards from the left pane of the Project Preferences dialog, the parameter options are as shown.



Use these options to select standards used in AutoCAD® Structural Detailing: a standard of the steel structure design and a drawing standard (drawings of steel structure elements). A selected drawing standard imposes designations (such as hatching symbols) that are valid in the country from which the standard originates.

Available standards:

- Structural standard
 - o Eurocode 3
 - o Polish code PN-90/B-03200
 - o French code CM66
 - o Russian code SNiP II-23-81
- Drawing standard (corresponding to a structural standard)
 - o ISO 129
 - Russian code GOST 21.101-97

2.1.10. Project Info

After you click Project Info from the left pane of the Project Preferences dialog, the parameter options are as shown.

Office:	
Investor:	Default User Name
Project name:	Test project title
Project No.:	Address: Company Address
Designed:	Revision: 01/01/2002
Verified:	Date: 01/02/2002
Note:	

Use the edit fields to enter general information about the project. These fields correspond to the variables used in cells of tables included in drawings and in printout headers:

Office - VAR_OFFICE
Investor - VAR_INVESTOR
Project name - VAR_PROJ_NAME
Project No. - VAR_PROJ_NUM
Address - VAR_ADDRESS
Designed - VAR_DESIGNER
Verified - VAR_VERIF
Revision - VAR_REV
Date - VAR_DATE

Additional variables: Scale - VAR_SCALE Page No. - VAR_PAGENO

See also:

Parameters - variables

2.2. Preferences

2.2.1. Preferences

Use this option to adopt basic parameters used in AutoCAD® Structural Detailing.

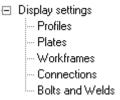
To begin working with preferences, open the Options dialog from:

- Menu: Steel / Preferences
- Ribbon: ASD Model / Settings / Preferences
- Toolbar: General / Preferences

• Command line: RBCS_PREF.

Click the Steel tab:

• In the left pane, use the selection tree (shown below) from to specify a program preference option

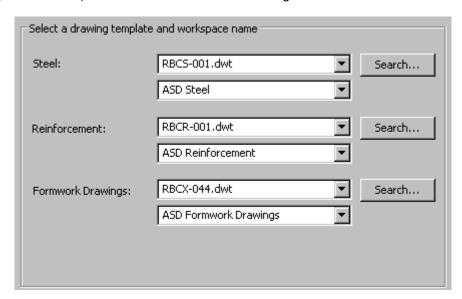


• In the right pane, specify parameters for the option selected on the left.

See also: Project preferences

2.2.2. General settings

After you click General settings from the left pane of the Structural Detailing tab (the AutoCAD® Options dialog), you can select a drawing template and a workspace name (such as ASD, ASD Classic) for AutoCAD® Structural Detailing modules.

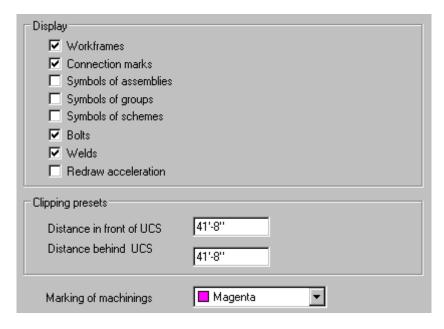


Available countries for which the working language may be selected:



2.2.3. Display settings

After you click Display settings from the left pane of the Steel tab, the parameter options are as shown.



Under Display, specify which elements to display in the drawing area:

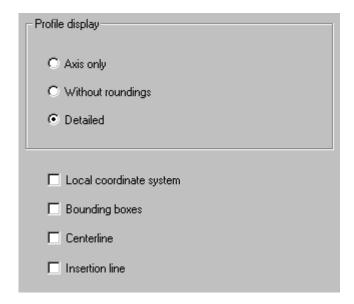
- Workframes
- Matching marks
- Connection marks
- Symbols of assemblies
- Symbols of groups
- Symbols of schemes
- Bolts
- Welds.

If Redraw acceleration is selected, then when an entire structure is displayed, a part of structure elements will not be shown (for example, connection details or connection symbols) due to an acceleration of model display on the screen. It is recommended to activate this option in the case of large structures.

Under Clipping presets, specify values for distances between clipping planes and UCS. For Marking of machinings, select a color of machinings made in elements of a structure model.

2.2.4. Profiles

After you click Profiles from the left pane of the Steel tab, the parameter options are as shown.

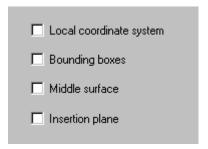


Define display parameters for bar profiles:

- For Profile display:
 - Axis only if this is selected, a profile (bar) will be displayed as a line connecting the beginning point and end point of the profile.
 - Without roundings if this is selected, a simplified view of a profile (without roundings) will be displayed.
 - o Detailed if this is selected, a profile (bar) will be displayed with all the details.
- Local coordinate system if this is selected, the local coordinate system will be displayed for all profiles defined in the project.
- Bounding boxes if this is selected, bounding boxes will be displayed for all profiles defined in the project.
- Centerline if this is selected, the bar axis will be displayed for all profiles defined in the project (regardless of which option has been chosen in the Profile display field).
- Insertion line if this is selected, the line of profile insertion will be displayed for all
 profiles defined in the project. This may be applied to present profile offsets.

2.2.5. Plates

After you click Plates from the left pane of the Steel tab, the parameter options are as shown.

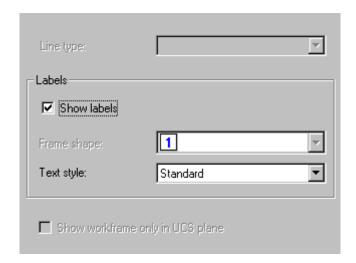


Specify display parameters for the defined plates:

- Local coordinate system if this is selected, the local coordinate system will be displayed for all plates defined in the project.
- Bounding boxes if this is selected, plate bounding boxes will be displayed for all plates defined in the project.
- Middle surface if this is selected, the middle surface will be displayed for all plates defined in the project.
- Insertion plane if this is selected, the insertion plane will be displayed for all plates defined in the project. This option may be applied in offset presentation.

2.2.6. Workframes

After you click Workframes from the left pane of the Steel tab, the parameter options are as shown.

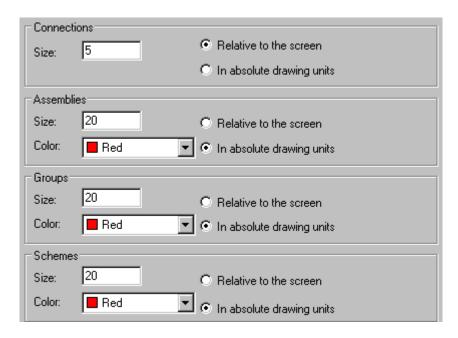


Specify display parameters for the defined workframes:

- The Line type list contains all line types available in the AutoCAD® project. Select a line type applied to lines of the workframes displayed in a structure model.
- Under Labels:
 - Show labels lets you switch the description of workframe lines on or off.
 - Frame shape specify the shape of descriptions of workframe lines (round, rectangular, or without frame).
 - Text style from the list of all text styles available in the AutoCAD®project, select a text style applied to descriptions of lines belonging to the workframes presented in a structure model.
- Show workframe only in UCS plane if this is selected, only these workframe lines which are positioned in the active local coordinate system are presented in the model.

2.2.7. Connections

After you click Connections from the left pane of the Steel tab, the parameter options are as shown.

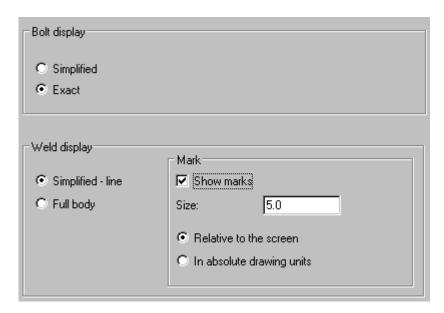


Specify display parameters for labels of defined connections (spheres), assemblies, and groups:

- Under Connections:
 - Size define the size of the connection symbol
 - o Relative to the screen if this is selected, the size is expressed as a percentage
 - \circ $\:$ In absolute drawing units if this is selected, the size is expressed in units used in $\:$ AutoCAD®
- Under Assemblies:
 - Color select a color for the assembly symbol
 - Size define the size of the assembly symbol
 - o Relative to the screen if this is selected, the size is expressed as a percentage
 - In absolute drawing units if this is selected, the size is expressed in units used in AutoCAD®
- Under Groups:
 - Color select a color for the group symbol
 - Size define the size of the group symbol
 - Relative to the screen if this is selected, the size is expressed as a percentage
 - In absolute drawing units if this is selected, the size is expressed in units used in AutoCAD®
- Under Schemes:
 - Color select a color for the scheme symbol
 - Size define the size of the scheme symbol
 - Relative to the screen if this is selected, the size is expressed as a percentage
 - In absolute drawing units if this is selected, the size is expressed in units used in AutoCAD®

2.2.8. Bolts and welds

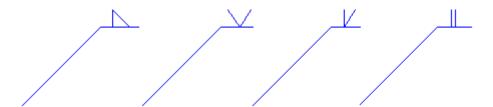
After you click Bolts and welds from the left pane of the Steel tab, the parameter options are as shown.



Specify display parameters for the defined bolts and welds:

- Under Bolt display:
 - Simplified if this is selected, bolts will be displayed as a line fragment (segment)
 - Exact if this is selected, bolts will be displayed with all shape details
- Under Weld display:

- Simplified line if this is selected, welds will be displayed as a line fragment (segment)
- Full body if this is selected, welds will be displayed with shape details displayed
- Under Mark:
 - Show marks use this to switch on/off the display of a weld mark. A mark shape also determines a weld type (see the drawing below which presents marks of several weld types). An extension line on which a weld mark is positioned is always perpendicular to the weld.



- Size define the size of a weld mark
- o Relative to the screen if this is selected, the size is expressed as a percentage
- In absolute drawing units if this is selected, the size is expressed in units used in AutoCAD®

Indicating a weld mark results in selecting a weld in a drawing.

3. OBJECT INSPECTOR

3.1. Description of Object Inspector

The Object Inspector is a tool that manages elements (objects) included in a project created in **AutoCAD® Structural Detailing**. By standard, the Inspector dialog displays in the program window, next to the graphical model definition field.

Important tasks carried out by the Object Inspector include:

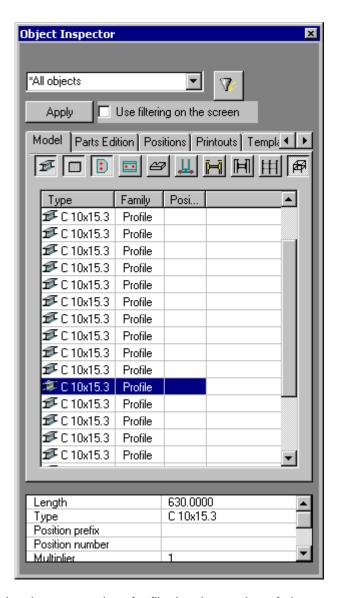
- Presenting the project contents in the appropriate order (sequence).
- Selecting elements subjected to actions of a selected command.
- Presenting and modifying project element properties (these may be single elements or whole objects).
- Activating the commands applied to a particular selection (it depends on the inspector mode).
- Defining, verifying and modifying groups.
- Verifying and modifying positions.
- Filtering model elements (objects).
- · Creating and managing drawing documentation of a project.

3.2. Object Inspector

The **Object Inspector** dialog is located to the left of the graphical model definition. The width of the dialog is adjustable to allow as much space as possible for the area of graphical model definition.

The **Object Inspector** dialog (shown below) is divided into three parts:

- Options that enable filtering model elements.
- Six tabs containing lists (set) of project elements depending on the stage of the design process (modeling / positions / printouts).
- A table presenting properties of selected objects.



At the top of the dialog there are options for filtering the number of elements displayed on the tabs. Filters enable the user to decrease the number of model elements shown in the graphical editor viewer.

The selection list contains the list of filters defined by the user in the current project.

Open the filter by clicking 1.

The filter list (defined in the Filter definition dialog) contains three default filters (which cannot be deleted):

- All objects cancels any filter and displays all objects defined in the current project on the list.
- Current selection an element list is limited to the elements that are currently selected in a model presented in the field of graphical model definition.
- *Hide current selection hi*des elements currently selected in a model presented in the field of graphical model definition (if there are no objects selected, then none of them can be hidden).

Click the **Apply** button to activate the filter. All objects defined by the filter are displayed on the list (even if they do not satisfy a filtering condition) until the next filtering operation.

Once the *Filter definition* dialog is closed, the list of filters defined is available in the *Object Inspector* dialog.

If the *Use filtering on screen* option is selected, the model graphical editor will display only the elements that are shown on the list in the Inspector dialog.

The dialog also includes the following tabs:

- Model
- Parts edition
- Positions
- Printouts
- Templates
- Structural Detailing Center.

The current list of objects defined in the project in *AutoCAD® Structural Detailing – Steel* display on successive tabs. Each of the tabs presents elements in different stages of a design process; therefore, each comprises different types of objects organized in a slightly different manner.

The bottom part of the dialog contains a table that displays properties of elements selected from the list. Table characteristics include:

- If one of the elements is selected, then the table presents all properties of a selected element
- If several objects of the same type are selected, then the table presents all the fields with
 properties relevant to the selected element type, however, only the values that are
 common for all the elements are presented; the remaining fields are empty (yet they may
 be changed)
- If several objects of different types are selected, the table presents only the fields with properties that are common for all object types.

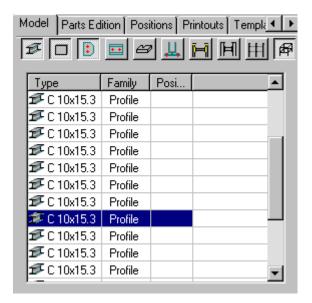
The first table column displays the name of a property (information), while the second table displays its current value.

The table with object properties designed to present these object properties that are important from the engineering viewpoint. It usually shows only basic information (properties). Not all the pieces of information displayed in the table can be changed.

See also: Object list

3.3. Model

Use this tab of the *Object Inspector* dialog to manage elements of a project model.



The following model objects defined in a structure are available on this tab:

- Profiles
- Plates

- Subparts
- User parts
- Connections
- Groups
- Workframes.

The following icons are located on the dialog box. The object types are presented on a list or may be excluded from presentation of objects.



- Plates

🖳 - Subparts (plates)

📃 - Subparts (profiles)

🖆 - User parts

- Connections

📕 - Groups

- Workframes.



NOTE:

The listed types of objects are filtered while being presented/hidden; the currently selected filter is applied.

When elements are selected from the list, they are highlighted in the graphical editor (the selection operation is interactive, all the elements chosen in the Inspector dialog are highlighted in the graphic editor: if an element is selected in the graphic editor, then the selection is also presented in the Inspector dialog).

An object list may be sorted by applying one of the following two methods:

- Double-clicking the table column header (the list can be sorted in ascending or descending order)
- Right-clicking the appropriate option in the context menu.



NOTE:

Sorting is always carried out separately for each element type.

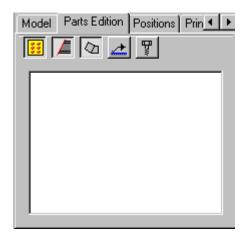
The context menu (accessed by right-clicking the Model tab), includes several options used to perform operations on selected objects:

- Select all select this command to select all elements presented on the list
- Deselect all select this command to cancel the selection of all elements on the list
- Remove select this command to remove selected elements from a project
- Zoom selection select this command to zoom-in on the selected elements in a structure view.
- Modify properties select this command to open the properties dialog (the dialog contents depend on a selected object type); this command is available only when selected elements are of the same type
- Group select this command to create a group that consists of selected elements; this
 command is available when several elements are selected
- Add to group select this command to add a selected element to an existing group
- Remove from group select this command to remove an indicated element from a selected group
- Change main part of group select this command to change the main part (main element) in a group; it is necessary to indicate an element that should be the main part of the group

- Ungroup select this command to split the group into single elements (objects); this
 command is available only when a group (or several groups) has been selected on the
 list
- Assign position select this command to assign a position to selected elements; when this command is activated, the *Manual Positioning* dialog opens
- Remove position select this command to remove positions from selected elements
- Auto positioning select this command to assign positions to selected elements; the **Automatic positioning** dialog displays.

3.4. Parts edition

This table presents edit operations performed on an element (e.g. cuttings, drills) as well as the contents of assemblies and groups. An element (or several elements) currently selected is presented in the form of a tree.



NOTE: The list of filter selections and the buttons allowing filter definitions are inaccessible on this tab. The *Current selection* filter is adopted by default.

Once the **Apply** button is clicked, selected objects are shown on the list. If a group is selected, then the group structure is displayed. If a single element is selected, then edit operations carried out on this element are presented.

The manner of presentation are changed by applying the following context menu options: *Show hierarchy of: Groups or Connections.*

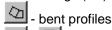
The following object types can be presented in the tree:

- Groups
- Connections
- Profiles
- Plates
- Subparts
- Drills
- Connectors (bolts, welds)
- Cuttings (fits)
- Notches.

The object types are presented on a list or excluded from the presented objects; the following icons located in the dialog serve this purpose:



- cuttings (fits)

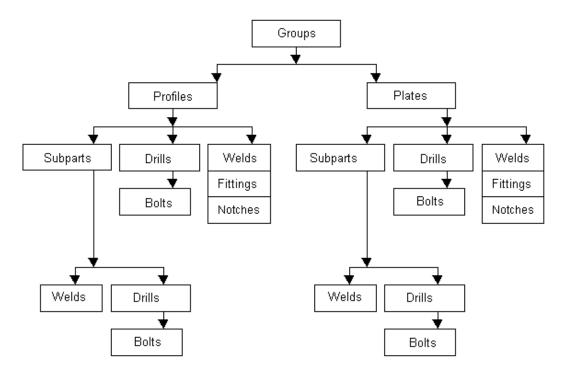


, T- connectors (welds, bolts).

All the listed icons are selected by default.

When elements are selected from the list, they are highlighted in the graphical editor (the selection operation is interactive, all the elements chosen in the Inspector dialog are highlighted in the graphic editor: if an element is selected in the graphic editor, then the selection is also presented in the Inspector dialog).

The hierarchy applied to present individual elements is illustrated in the figure below. There is a separate hierarchy for each connection.

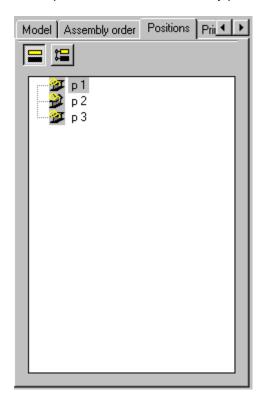


The context menu (accessed by right-clicking the *Parts Edition* tab), includes several options used to perform operations on selected elements:

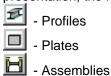
- Remove select this command to remove selected elements from a project
- Zoom selection select this command to zoom-in on the selected elements in a structure view
- Modify properties select this command to open the properties dialog (the dialog contents depend on a selected object type); this command is available only when one element is selected on the list
- Assign position select this command to assign positions to selected elements; activating
 this command opens the *Manual Positioning* dialog.
- Remove position select this command to remove positions from selected elements.

3.5. Positions

This tab presents a list of defined positions; the list is sorted by prefixes and numbers.



The position types shown below are presented on the list or may be excluded from presentation; the following icons, located in dialog, are used for this purpose:



H - Groups

- User parts

Below are three selection options (*Parts*, *Families*, *Groups*) used to activate a mode of presentation (sorting) of positions.

With the Parts option selected

In this mode elements are shown in one level, grouped by position types in the order as follows (the first order criterion for individual parts is a prefix, the next one is a number), in the alphabetical order of position designations:

- 1. Profiles
- 2. Plates
- 3. Assemblies
- 4. Groups
- 5. User parts
- 6. Schemes

With the Families option selected

In this mode elements are presented in the form of a tree with the following levels:

- 1. Position type; the order of the items: assemblies, profiles, plates, user parts
- 2. Family (in the alphabetical order)
- 3. Within a family positions are presented in the alphabetical order of designations

With the Groups option selected

This mode offers multi-level presentation of elements; it is reflected in this mode which parts are assigned to which assemblies and groups. Positions are presented in the form of a tree with the following levels:

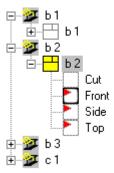
1. Position types; the order of the items: groups, assemblies, profiles, plates, user parts, schemes

For the types Groups and Assemblies successive sub-levels are created:

for groups: elements included in a given group

for assemblies: single parts included in a given assembly.

This tab presents a list of all user-defined positions; the structure of positions and documents is shown in a form of a tree (see the drawing below).



The icon of a document included on the list is presented as follows:

- in yellow the document is active on the edition layout
- day as a red slash the element of a structure model for which the document has been prepared, has changed
- in gray the document has been exported to a separate dwg file.

If the icon of a view provided on the list is shown in bold, it is an active view.

The context menu (accessed by right-clicking the *Positions* tab), includes several options used to perform operations on selected positions:

- Delete select this command to remove the selected positions from a project
- Select positions use this command to select all positions
- Attach document use this command to select a printout template; the template is saved on the Templates tab
- Update documents select this command to update the documents available for a position
- Delete documents select this command to delete documents available for a position
- Automatic drawings select this command to automatically generate drawings for a selected position; after running this command, the Automatic drawing generation dialog displays.

3.6. Printouts

This tab enables management of printouts in *AutoCAD® Structural Detailing*; it presents the list of all printouts defined in the *AutoCAD® Structural Detailing* project.

Printouts are presented together with a set of views. The printout list contains all the printouts, even those which do not include any views. The structure of user-defined printouts and views is shown in a form of a tree. Due to logical reasons, views are placed in a printout, however, the tree also includes an intermediate level, so that it is obvious to which document given views belong. If the printout layout is active, then the icon of a printout corresponding to the active printout layout displays in red.

When printouts provided on the list are selected, only elements of one printout may be selected at a time - it is impossible to select elements of two different printouts. Selection of the printout in the dialog is synchronized with the graphic editor - an appropriate drawing displays.

The top part of the dialog contains two icons that enable presentation of the existing positions in the tree in the following modes:

- model mode
- drawing mode.

The context menu (accessed by right-clicking the *Printouts* tab), includes several options used to perform operations on selected printouts:

- Delete select this command to remove selected printouts from a project
- Change name choose this command to change the name of a highlighted printout (the name is entered to the command line)
- Add printout select this command to add an empty printout to the project (the name is entered to the command line)
- Activate select this command to activate a selected printout
- Unload printout select this command, to make a selected printout no longer active
- Save printout select this command to save a selected printout in a *.DWG format file
- Update select this command to update a selected printout after making modifications in positions
- Update all printouts select this command to update all printouts after making modifications in positions.

3.7. Templates

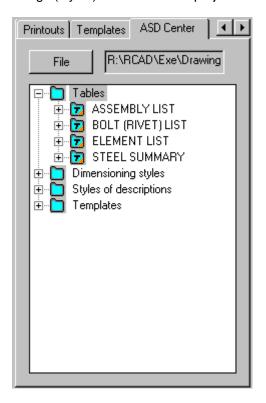
This tab enables management of printout templates in *AutoCAD® Structural Detailing*; it presents the list of all printout templates defined in the *AutoCAD® Structural Detailing* project; the template structure is presented in a form of a tree. The icon of an active template (in the template layout) displays in yellow.

The context menu (accessed by right-clicking the *Templates* tab), includes several options used to perform operations on selected templates:

- Delete select this command to remove selected templates from a project
- Change name choose this command to change the name of a highlighted template (the name is entered to the command line)
- Activate choose this command to activate the highlighted template
- Add view select this command to add a view to an existing template
- Change style choose this command to change the style in a selected template
- Add document template select this command to add a template to a selected view.

3.8. Structural Detailing Center

This tab enables copying settings (styles) between user's projects.



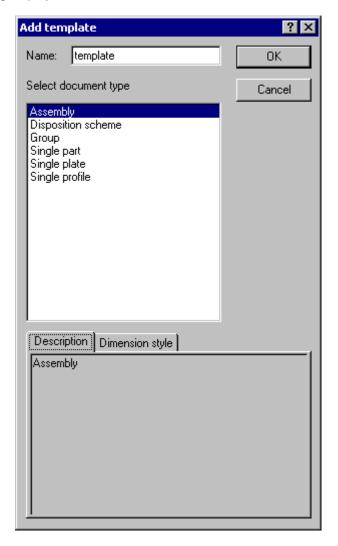
After clicking **File** and selecting a file with an earlier-saved project, the Inspector dialog displays all the styles defined in the selected project, which may be used in the current project. After highlighting a selected style, right-click the *Copy* command; the selected style is available in the current project.

3.9. Add template

The dialog is used to define a printout template for drawings of positions. There are three ways to access this option from:

- Menu: Steel / Edit Drawings / Drawing Template Designer
- Ribbon: ASD Drawings / Generate Drawings / Drawing Template Designer
- Toolbar: Edit Drawings / Drawing Template Designer
- Command line: RBCS ADDTEMPLATE.

The following dialog displays:



The dialog defines a template for created drawings. For steel, select one of the allowable document types concerned with the position type: assembly, disposition scheme, group, single part, single plate or single profile.

After providing the name of a template and clicking **OK**, the template is added to the list of available templates located on the *Templates* tab.

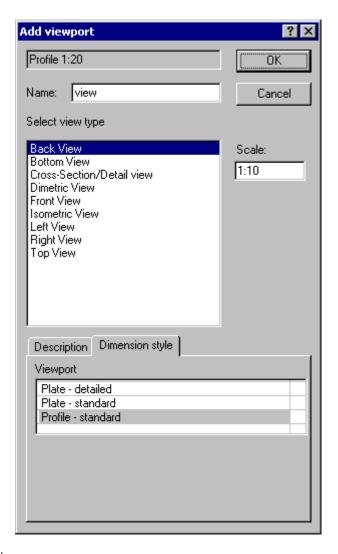
In the bottom of the dialog, the following tabs are included:

- Description (this tab presents description of a selected position type),
- *Dimension style (*this tab presents available dimension styles for a selected position type).

3.10.Add view

Once the template name and type are specified, you define views included in a given template. The *Add viewport* dialog is available after right-clicking the *Add view* option on the *Templates* tab in the *Object inspector* dialog.

The following dialog displays:



To place a single view:

- 1. Name of a created view (a name of the currently-applied template is specified)
- 2. Select a view (projection) of a structure element
- 3. Specify the scale for a selected view.

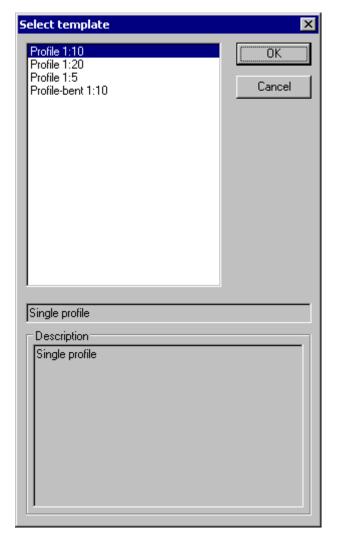
After clicking **OK** within the template layout, the placement of a view in a template (not yet in a final drawing) should be indicated.

Repeat these steps to place successive views in a template.

3.11. Select template

Once a position is defined, you select a drawing template for it. The **Select template** dialog is available after right-clicking the **Attach document** option on the **Positions** tab in the **Object inspector** dialog.

The following dialog displays:



In the dialog, choose a template to be applied for generating a drawing from the list of available (defined) templates. Click **OK** to assign the template to a selected position. After selecting the *Add to current Printout* option (context menu on the *Positions* tab), a created document is placed in a final drawing (on the Layout tab).

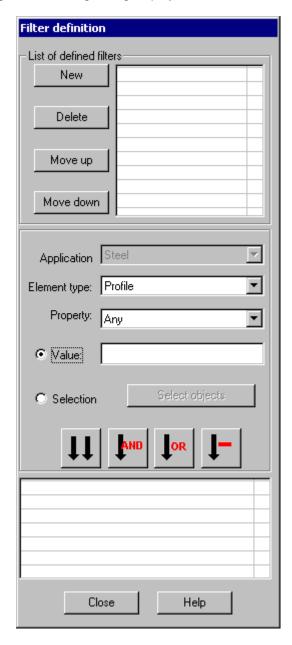
3.12.Templates/views - steps to be followed while creating a printout

To make a final drawing (in the case of steel structure elements):

- 1. Define a template:
 - In the Add template dialog, specify the name and select the type of structure element (single part).
 - Determine views (projections) that are to be included in a template
- 2. In the *Positioning* (manual or automatic) dialog, define positions for structure elements.
- 3. In the **Select template** dialog, assign a template to a selected position.
- 4. Place documents in a final drawing (printout).

3.13. Definition of filters

You define filters in the *Filter definition* dialog. Access the dialog by clicking , in the Object inspector dialog. The following dialog displays:



Filter definition are based on types of model elements and their properties. A defined filter is identified by a name displayed on the filter selection list located in the Object Inspector dialog. The *Filter definition* dialog allows you to define a new filter or modify an existing filter (except for the two default filters). User-defined filters are deleted from the list by clicking the Delete button.

The top part of the dialog contains the list of all the filters defined by the user in a project. A selected (highlighted) filter is an active filter (i.e. a filter selected when the dialog was opened). The bottom part of the dialog includes the field for defining the selected filter. The steps taken while defining and modifying a filter are similar.

To define a new filter:

- 1. Click the **New** button in the top part of the dialog. The **New filter name** dialog displays.
- 2. Enter a name for the filter in the edit field.

Note: If the *Inherit definition* option is on, then a new filter will inherit a definition from the recently-selected filter (which means that the expression of the recently-selected filter will be entered on the list located in the bottom part of the *Filter definition* dialog); if the *Inherit definition* option is off, then a new filter will be created without any definition of expression (the list located in the bottom part of the *Filter definition* dialog box will be empty).

3. Close the **New filter name** dialog, the defined filter displays on the list contained in the **Filter definition** dialog box.

After providing a filter name, the user may determine the expression defining a filter. From the *Type* drop-down list, select a type of element that is to be included in the defined filter. The contents of the *Properties* list (see also List of available elements, properties and values) will change depending on the element type selected.

Select a filtering criterion from the *Properties* list. Criterion depends on the selected element type.

Depending on the selected type of property, determine the list of selected property values in the *Value* edit field. This field allows entering values that appear in the expression defining a filter.

If the *Properties* and *Value* field are left blank, all the elements of the selected type (all values) will be included in the filter defined.

For some of the properties (e.g. length), define a range of values in the *Value* field; the following syntax is applied:

- Single values separated with a semicolon e.g. 1; 2; 3; 10; only the elements listed will be included in a filter.
- Range of values defined as follows: 1-100.
- Values greater than and less than: -100; 10-; (respectively).

Apart from definition of values, it is also possible to define graphically the elements that are included in a filter; once the *Selection* option is selected. Once the **Select objects** button becomes accessible; clicking this button closes the dialog, whereas the cursor is in the selection mode. Right-click the structure elements (or click the ENTER key) to display the elements in the dialog.

To include an expression or selection in a filter, it is necessary to use one of the buttons shown below:

- Clicking this button replaces all the expressions defined by the expression currently determined.
- Clicking this button adds the current expression to expressions already defined in a filter (the AND operator is added in front of the current expression).
- Clicking this button adds the current expression to expressions already defined in a filter (the OR operator is added in front of the current expression).
- Clicking this button adds the current expression to expressions already defined in a filter (the AND NOT operator is added in front of the current expression).

All expressions added to a defined/modified filter are displayed in the bottom part of the dialog with appropriate operators. Each expression is entered into a separate line; lines are separated with logical operators referring to expressions below.

You cannot modify the contents of a single expression in a filter; however, it is possible to delete lines from definitions of expressions in a filter. Delete lines by right-clicking on one of the following commands within the bottom part of the *Filter definition* dialog.

- Delete line once this command is selected, a highlighted line is deleted.
- Delete all once this command is selected, a whole filter definition is deleted.

Each of the filters can be modified at any moment while working in a project. The list of filters defined can be seen in the top part of the *Filter definition* dialog. A filter defined in the project is deleted from the list of available filters by highlighting a line with a filter name and clicking **Delete**. The order of filters defined in a project can be changed on the filter list by applying the buttons: **Move up** or **Move down**. The highlighted filter is moved one position up or down on the list of defined filters.

3.14. How to define a filter

Example of how to define a filter allowing a selection of columns made of RSJ 102 \times 102 \times 23.07 and RSJ 76 \times 76 \times 12.65 sections:

- Click the New button; in the New filter name dialog, enter the name of a defined filter.
- Close the New filter name dialog (the Inherit definition option is switched off); a new filter displays in the Filter definition dialog.
- Select Section as Type, Section type as Properties; on the Values list select lines with sections RSJ 76 x 76 x 12.65 and RSJ 102 x 102 x 23.07 (the lines are highlighted).
- Add an expression by pressing the $\downarrow \downarrow$ button; the following expression displays: Sections (type) = RSJ 76 x 76 x 12.65; RSJ 102 x 102 x 23.07.
- Select Section as Type; Family as Properties; on the Values list select the Column family.
- Add an expression by pressing the ↓AND button; the following expression displays: Sections (type) = RSJ 76 x 76 x 12.65; RSJ 102 x 102 x 23.07 and Sections (family) = Column.

The operation above completes the filter definition.

3.15. Objects and their properties

The Object Inspector includes:

Model elements (physical elements)

- Profiles
- Plates (single plates)
- User parts
- Subparts (such as plates, profiles, ribs, shear plates)
- Welds
- Bolts
- Connections
- Groups
- Assemblies a set of parts (profiles or plates) grouped automatically based on the connectors with the 'workshop' attribute existing between them; in practice, these are elements sent as a whole from a workshop to a construction site
- Workframes.

Model elements (nonphysical elements)

- Drills
- Openings
- Cuttings/fits
- Notches.

Drawing elements

- Positions
- Documents (sets of views)
- Views
- Printouts.

Below, the table of available elements, properties and values display.

Objects	Properties All properties for objects	Values Values appropriate for object properties
Profiles	Profile type Family Material Length Position prefix Position number Subpart	Profiles available in the current project Families intended for the "profiles" type available in the current project Materials set in preferences Real positive value Position prefixes used in the current project Integral positive number Yes / No depending on that whether a selected profile is a subpart
Plates	Plate thickness Family Material Type Position prefix Position number Subpart	Thickness of a project or any real positive value Families available in the current project intended for the "plate" type Materials set in preferences Rectangular, circular, user-defined Position prefixes used in the current project Integral positive number Yes / No depending on that whether a selected plate is a subpart
Subparts		Properties as for a profile or plate depending on the subpart type
Connections	Connection type	Available connection types: (e.g. endplate)
Groups	Name Position prefix Position number Subpart	Text Position prefixes used in the current project Integral positive number Yes / No depending on that whether a selected profile is a subpart
Workframes	Workframe name Workframe type	Workframe names available in the current project Box, wedge, prism
Positions	Prefix Number Level of positioning	Position prefixes used in a project Position numbers defined in the <i>edit field</i> Single part, assembly, group
Printouts	Number Name Format Positions	Numbers defined in the <i>edit field</i> Drawing names available in the current project Formats used in the current project Positions defined in the current project

4. TYPICAL STRUCTURES

4.1. Typical structures

Use this option to define typical (parametric) structures.

To begin defining typical structures, go to:

- Menu: Steel / Parametric structures
- Ribbon: ASD Model / Parametric structures
- Toolbar: Parametric structures.

Typical structures available in the current version of AutoCAD® Structural Detailing:

Parametric structures (Steel menu > Parametric structures), ribbon (ASD - Model > Parametric Structures) or Parametric structures toolbar

- Multi-bay frame M
- Roof truss
- Stairs 🗳
- Stairs spiral
- Staircase <
- Bracing X
- Ladder |
- Railing #
- Cage 🖽
- Brace 3
- Grate spreading 4
- Purlin spreading 4

Plate girders (Steel menu > Plate girders, ASD - Model > Elements ribbon or Plate girder; General toolbar)

- Castellated beam
- Column (plate girder)
- Beam (plate girder)
- Multisegment beam (plate girder) ...

After you select a structure, an additional dialog displays in which you can define parameters of a selected steel structure. The contents of this dialog depend on a selected structure type or structure element.

See also:

Description of the macros for generation of steel structure elements Description of the macros for generation of connections of steel structure elements

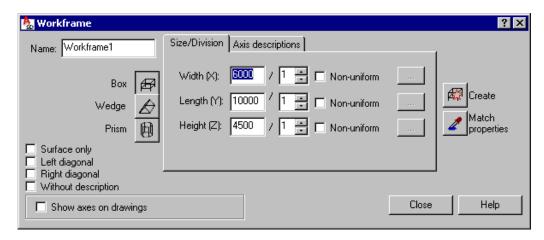
5. WORKFRAME

5.1. Workframe

Use this option to determine a workframe that facilitates definition of a structure model in 3D space.

To begin working with workframes, open the Workframe dialog from:

- Menu: Steel / Workframes / Create workframe
- Ribbon: ASD Model / Elements / Workframes
- Toolbar: General / Workframes
- Command line: RBCS WF.



The Workframe dialog has 3 main parts:

- On the left, define basic information for a workframe
 - o Name
 - Typeboxwedge
 - prism
- Use the options at the bottom of the dialog:
 - Surface only if this is selected, only external lines of a workframe are created. If this is not selected, all workframe lines are created.
 - Left diagonal if this is selected, additional lines (diagonals of a quadrangle area inclined to the left) are also created.
 - Right diagonal if this is selected, additional lines (diagonals of a quadrangle area inclined to the right) are also created.
 - Without description if this is selected, descriptions of axes are not provided. If this is not selected, workframe axes will be described.
 - Show axes in drawings if this is selected, axes of a defined workframe are displayed in drawings. If this is not selected, workframe axes play only an auxiliary role in modeling and are not shown in any drawing.
- Select a tab at the middle of the dialog Size/Division or Axis descriptions; their contents depend on a selected type of a workframe.
- · Use the icons on the right
 - $rac{M}{2}$ creates a workframe based on parameters determined in the dialog.
 - is used to adopt (inherit) parameters from a workframe that is already defined.

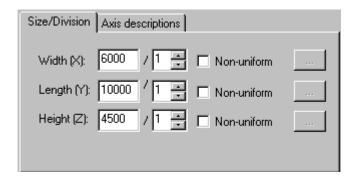
When the Workframe dialog displays, it displays the workframe type most recently defined and the associated parameters.

See also:

Example of definition of a workframe

5.2. Box

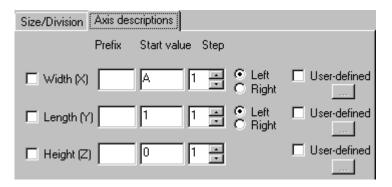
After you select Box as a workframe type, the Workframe dialog contains 2 tabs: Size/Division and Axis descriptions.



Options on the Size/Division tab:

- Width (X) define the width of a workframe, and the number of divisions along the workframe width
- Length (Y) define the length of a workframe, and the number of divisions along the workframe length
- Height (Z) define the height of a workframe, and the number of divisions along the workframe height

If the Non-uniform option next to the options is not selected, a workframe will be generated automatically based on the parameter values defined in the edit fields. If Non-uniform is selected, the edit fields become inaccessible. The (\dots) button becomes available; click it to open the Lines distribution dialog where you can define the position of successive workframe axes.



Options on the Axis descriptions tab:

- Width (X) define an axis description along the workframe width. If this is selected, the
 description will be generated; if it is not selected, the axis description will not be
 generated.
- Length (Y) define an axis description along the workframe length. If this is selected, the description will be generated; if it is not selected, the axis description will not be generated.
- Height (Z) define an axis description along the workframe height. If this is selected, the description will be generated; if it is not selected, the axis description will not be generated.

For each of the options, there are 3 fields:

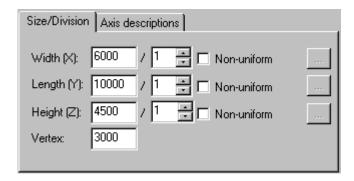
- Prefix define a prefix of workframe axes for the automatic description of workframe axes
- Start value define a start value for workframe axes for the automatic description of workframe axes
- Step define a value of increment for successive workframe axes for the automatic description of workframe axes

For width (X) and length (Y), you can choose to locate the description on the right or on the left of a workframe line.

If the User-defined option next to the options is not selected, a workframe description will be generated automatically based on the parameter values defined in the edit fields. If User-defined is selected for any of the options, the edit fields for that option become inaccessible. The (...) button becomes available; click it to open the Lines description dialog where you can define the description of successive workframe axes.

5.3. Wedge

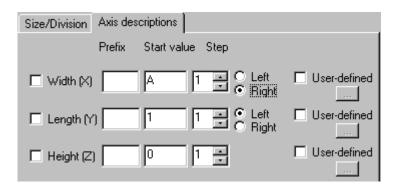
After you select Wedge as a workframe type, the Workframe dialog contains 2 tabs: Size/Division and Axis descriptions.



Options on the Size/Division tab:

- Width (X) define the width of a workframe, and the number of divisions along the workframe width
- Length (Y) define the length of a workframe, and the number of divisions along the workframe length
- Height (Z) define the height of a workframe, and the number of divisions along the workframe height
- Vertex define a coordinate of the vertex position

If the Non-uniform option next to the options is not selected, a workframe will be generated automatically based on the parameter values defined in the edit fields. If Non-uniform is selected, the edit fields become inaccessible. The (...) button becomes available; click it to open the Lines distribution dialog where you can define the position of successive workframe axes.



Options on the Axis descriptions tab:

- Width (X) define an axis description along the workframe width. If this is selected, the
 description will be generated; if it is not selected, the axis description will not be
 generated.
- Length (Y) define an axis description along the workframe length. If this is selected, the description will be generated; if it is not selected, the axis description will not be generated.
- Height (Z) define an axis description along the workframe height. If this is selected, the description will be generated; if it is not selected, the axis description will not be generated.

For each of the options, there are 3 fields:

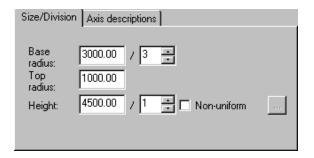
- Prefix define a prefix of workframe axes for the automatic description of workframe axes
- Start value define a start value for workframe axes for the automatic description of workframe axes
- Step define a value of increment for successive workframe axes for the automatic description of workframe axes

For width (X) and length (Y), you can choose to locate the description on the right or on the left of a workframe line.

If the User-defined option next to the options is not selected, a workframe description will be generated automatically based on the parameter values defined in the edit fields. If User-defined is selected for any of the options, the edit fields for that option become inaccessible. The (...) button becomes available; click it to open the Lines description_dialog where you can define the description of successive workframe axes.

5.4. Prism

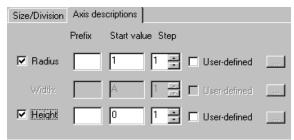
After you select Prism as a workframe type, the Workframe dialog contains 2 tabs: Size/Division and Axis descriptions.



Options on the Size/Division tab:

- Base radius define a radius of the circle circumscribed on the prism base, and the number of radial divisions
- Top radius define a radius of the circle circumscribed on the prism top
- Height define the height of a workframe, and the number of divisions along the workframe height

If the Non-uniform option next to Height is not selected, a workframe will be generated automatically based on the parameter values defined in the edit fields. If Non-uniform is selected, the edit fields become inaccessible. The (...) button becomes available; click it to open the Lines distribution dialog where you can define the position of successive workframe axes.



Options on the Axis descriptions tab:

- Radius define an axis description along the workframe radius. If this is selected, the
 description will be generated; if it is not selected, the axis description will not be
 generated.
- Width not available
- Height define an axis description along the workframe height. If this is selected, the
 description will be generated; if it is not selected, the axis description will not be
 generated.

For each of the options, there are 3 fields:

- Prefix define a prefix of workframe axes for the automatic description of workframe axes
- Start value define a start value for workframe axes for the automatic description of workframe axes
- Step define a value of increment for successive workframe axes for the automatic description of workframe axes

If the User-defined option next to the options is not selected, a workframe description will be generated automatically based on the parameter values defined in the edit fields. If User-defined is selected for any of the options, the edit fields for that option become inaccessible. The (...) button becomes available; click it to open the Lines description dialog where you can define the description of successive workframe axes.

5.5. Lines distribution

If the Non-uniform option is selected for any of the options on the Size/Division tab of the Workframe dialog, the edit fields become inaccessible. Click the (...) button to open the Lines distribution dialog, which provides the table that lets you determine the position of the successive workframe axes.

To define the position of workframe axes:

- 1. Define spacing between successive workframe axes, or define the coordinate of the workframe axis (calculated from the workframe beginning).
- 2. Perform the above operation for the successive workframe axes.
- 3. Click OK.

5.6. Lines description

If the User-defined option is selected for any of the options on the Axis descriptions tab of the Workframe dialog, the edit fields become inaccessible. Click the (...) button to open the Lines description dialog, which provides the table that lets you determine descriptions of the successive workframe axes.

To define descriptions of workframe axes:

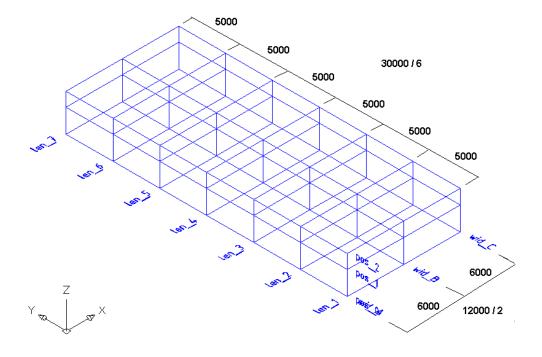
- 1. Define the description of the first axis.
- 2. Move to the next table line and define the description of the second axis.
- 3. For some descriptions (such as width X and length Y), select the location of the axis description, either on the right or on the left of the workframe line.
- 4. Perform the above operation for the defined workframe axes.
- 5. Click OK.

5.7. Example of definition of a workframe

To start definition of a workframe, do as follows:

- 1. Click the Workframes option
- 2. In the Workframe dialog:
 - For Name, enter Workframe 3D Workshop.
 - For workframe type, select Box.
 - Verify that Surface only, Left diagonal, Right diagonal, and Without description are cleared.
 - · Select Show axes in drawings.
- 3. On the Size/Division tab, define workframe dimensions with respect to the system
 - Width (X): 12000 / 2
 - Length (Y): 30000 / 6
 - Height (Z): select Non-uniform and click the (...) button.
 - In the Line distribution (length) dialog, in the Coordinate column, enter **5000**, **7500**, **9000**, and then click OK.
- 4. On the Axis descriptions tab, select descriptions for all the dimensions, and choose the following descriptions for individual workframe axes:
 - Width (X): Prefix: wid_, Start value: A, Step: 1, left
 - Length (Y): Prefix: len_, Start value: 1, Step: 1, left
 - Height (Z): Prefix: pos_, Start value: 0, Step: 1
- 5. Click Create (located on the right of the Workframe dialog).
- 6. Specify the workframe origin and determine the direction of the workframe X axis: direction of the X axis of the global coordinate system (NOTE: the direction of the X axis of the workframe does not need to be compatible with the X axis of the global coordinate system).

The generated workframe is illustrated in the drawing below.



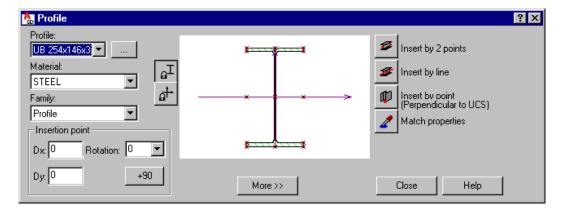
6. PROFILES

6.1. Profiles

Use this option to define a member with a selected profile.

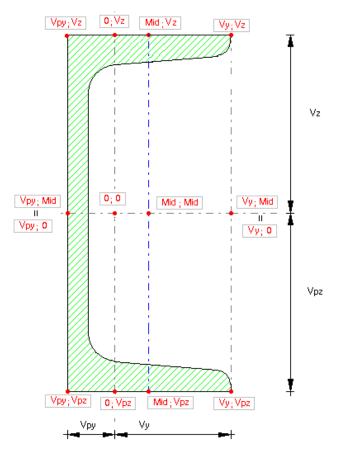
To begin working with profiles, open the Profile dialog from:

- Menu: Steel / Profiles
- Ribbon: ASD Model / Elements / Profiles
- Toolbar: General / Profiles
 Command line: RBCS PROFILE.



The Profile dialog has 3 main parts:

- On the left, you can specify basic information for a profile of a defined member:
 - Profile select a profile for a defined member (the most recently defined member profile is displayed). Click the (...) button to open the Profile List dialog, in which profiles from the available databases may be added to the list of available profiles.
 - o Material select the type of a material assigned to a defined member
 - Family select a family (group) of members to which a defined member will be assigned
 - Insertion point define coordinates of the member shift (Dx and Dy offsets) and the rotation of a member profile about the member longitudinal axis (select a value from the list or enter one into the Rotation field). The offset denotes the shift of a profile center with respect to the insertion line
- In the middle is a graphical field that displays a selected profile of a defined member
 positioned in relation to the insertion line (under the graphical field, characteristic
 distances of the profile fibers from the profile axis are displayed). You can define
 offsets graphically, by using the cursor to shift a profile with respect to the axis or by
 specifying characteristic points of a profile.
 - You can also define relative offsets (with respect to values related to the cross-section and characteristic points of a profile). Location of section characteristic points (their number depends on the section type and shape) may be determined by specifying certain values connected directly with the cross-section (these do not have to be absolute values). Characteristic points include:
 - (0;0) center of gravity of a section
 - Points presented in the drawing below whose coordinates are determined by the values Vy, Vpy, Vz, Vpz, Mid (they define a relative location of a point in the cross section)



Profile characteristic points are marked in red in the dialog (in the drawing of a selected profile).

After you click on a characteristic point of a profile, the profile is translated and the edit fields Dx and Dy in the Insertion point field are populated automatically.

Under the schematic drawing, there is the More >> (or Less <<) button, which opens (closes) an additional field where you can specify:

- a surface finishing type for a profile (for example, hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat). A default type of surface finishing can be selected in the Project preferences dialog on the Materials tab; this dialog also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types.
- o profile properties (area, moments of inertia, and so on) see the description of the Profile list dialog. After you right-click in this field, the context menu displays:
 - Select All selects the contents of the edit field
 - Copy copies the contents of the edit field to the Clipboard
 - Paste pastes the Clipboard contents to the edit field

The following icons are also available in this part of the dialog:

- blocks the position of axes in the graphical field and lets you shift a profile
- blocks the position of a profile in the graphical field and lets you shift profile axes
 - Use the icons on the right to define the profile insertion:
 - 2 points
 - 🌌 line
 - 1 point (a member will be defined as perpendicular to UCS)
- Match properties used to adopt (inherit) parameters from a defined member profile.

When the Profile dialog displays, it displays the member profile parameters defined most recently.

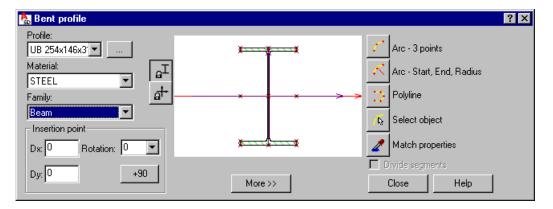
See also: Example of definition of a user-defined section

6.2. Bent profiles

Use this option to define a bent bar of a selected profile.

To begin defining a bent profile, open the Bent profile dialog from:

- Menu: Steel / Bent profiles
- Ribbon: ASD Model / Elements / Bent profiles
- Toolbar: General / Bent profiles
- Command line: RBCS_BENT.

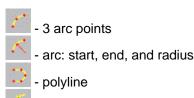


The Bent profile dialog has 3 main parts:

- On the left, you can specify basic information for the profile of a defined bar:
 - Profile select a profile of a defined bar (the most recently defined bar profile displays). Click the (...) button to open the Profile List_dialog, where profiles from the databases accessible in the software may be added to the list of available profiles.
 - Material select the type of material assigned to a defined bar
 - o Family select a bar family (group), to which a defined bar will be assigned
 - Insertion point define coordinates of the bar offset (Dx and Dy), and the rotation
 of the bar profile about the bar longitudinal axis (select a value from the list or
 enter a value in the Rotation field). The offset denotes translation of the profile
 center with respect to the insertion line.
- In the middle is a graphical field that displays a selected profile of a defined bar
 positioned in relation to the insertion line (under the graphical field, characteristic
 distances of the profile fibers from the profile axis are displayed). You can define
 offsets graphically, by using the cursor to translate a profile with respect to the axis or
 by specifying characteristic points of a profile.
 - Under the schematic drawing, there is the More >> (or Less <<) button, which opens (closes) an additional field where you can specify:
 - a surface finishing type for a profile (for example, hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat). A default type of surface finishing can be selected in the Project preferences dialog on the Materials tab; this dialog also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types.
 - o profile properties (area, moments of inertia, and so on) see the description of the Profile List dialog. After you right-click in this field, the context menu displays:
 - Select All selects the contents of the edit field
 - Copy copies the contents of the edit field to the Clipboard
 - Paste pastes the Clipboard contents to the edit field

The following icons are also available in this part of the dialog:

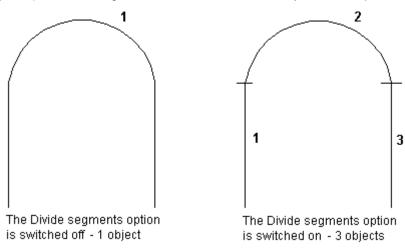
- blocks the position of axes in the graphical field and lets you translate a profile
- blocks the position of a profile in the graphical field and lets you translate profile axes
- Use the icons on the right to define the bar insertion:



- select object

Match properties - used to adopt (inherit) parameters from a defined bar profile.

Below the icons is the Divide segments option - if this is not selected, a bent profile is defined as one object (despite that it may consist of several elements, such as a polyline composed of arc, segment, and arc). If this is selected, the bent profile is divided into separate objects when the type of profile parts changes (a polyline composed of arc, segment, and arc is divided into 3 separate parts: 2 arcs and a segment). The drawing below illustrates schematically how this option works.



When the Bent profile dialog displays, it displays parameters of the most recently defined bar profile.

6.3. Example of definition of a user-defined section

The example below presents the definition of a section of a steel flat bar that may be used in definition of a structure model. To define a user-defined section (bar section):

- Applying AutoCAD® options:
 - Draw a rectangle (Draw menu > Rectangle) with the dimensions 590x4050 (see the drawing below).
 - Select a region (Draw menu > Region) and indicate the rectangle drawn.

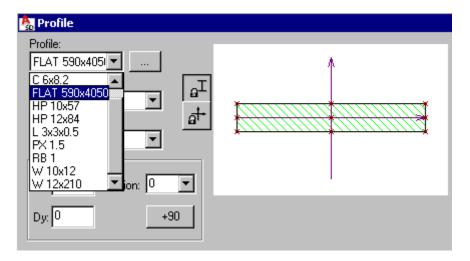


- Applying AutoCAD® Structural Detailing Steel options:
 - Click the User sections
 - Specify the defined region (rectangle).
 - Select to save the section in the current project (the Locally option in the command line).
 - Specify a section name (4 letters and designations describing profile dimensions, such as FLAT 590x4050).
 - o Define a profile shape code.



Section codes are adopted according to designations used in Robot; code descriptions are provided in Robot help.

Thus-defined section of a profile will be added to the list of available profiles (in the Profile dialog, a part of which is shown in the drawing below). The section will be available only in the current project.



If Database is selected, use the Saving section to database dialog to save a profile to a database:

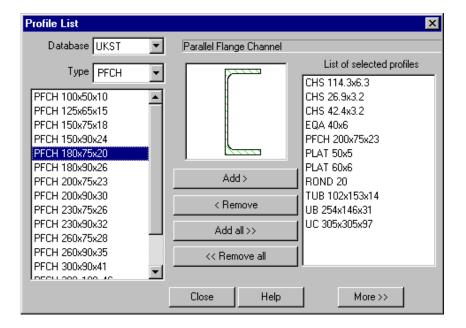
- Specify a name, a maximum of 4 letters (digits are not allowed).
- Define dimensions of the profile (real numbers defining information about the profile)
 - o If 1 dimension is given, specify Dimension 1
 - If 2 dimensions are given, specify Dimension 1 and Dimension 3

See also:

List of profile shape codes

6.4. Profile list

The Profile list dialog displays after you click the (...) button in the Profile dialog.



Using this dialog, you can add profiles from databases accessible in the program to the list of profiles available in AutoCAD® Structural Detailing - Steel. To add a profile to the list of available profiles:

- Select a profile folder in the Database field, such as AISC (American Hot Rolled Shapes).
- Select a profile type.
- Select a profile in the left panel.
- Click Add >.

Clicking Add all >> results in adding all the profiles included in the left panel.

Clicking < Remove deletes a selected profile from the list of available profiles. Clicking << Remove all results in deleting all the profiles provided on the list of available profiles (in the right panel) except for the profiles that have been used in the project.

The Remove option is unavailable for profiles that have already been applied in a project. The bottom right corner of the dialog contains the More >> (or Less <<) button, which opens an additional part of the dialog with information on basic section properties (dimensions of the cross-section, area, moments of inertia, section modulus for calculation of torsion stresses, elastic section moduli (bending), mass). This additional part of the dialog is closed when you click Less >>.

7. PLATES

7.1. Plates

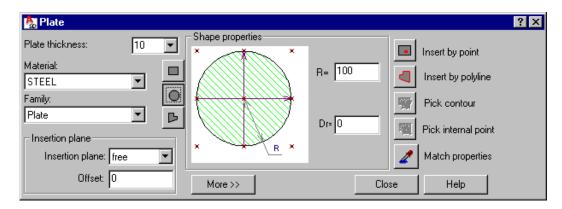
Use this option to define a plate.

To begin defining a plate, open the Plate dialog from:

Menu: Steel / Plates

• Ribbon: ASD - Model / Elements / Plates

Toolbar: General / Plates
Command line: RBCS PLATE.



The Plate dialog has 3 main parts:

- On the left, you can specify basic information for a defined plate:
 - Plate thickness select a value of plate thickness from the values on the list or enter a value into the edit field
 - Material select the type of material assigned to a defined plate
 - Family select a family (group) of plates to which a defined plate will be assigned Use the icons to define the plate shape:
 - rectangular plate
 - circular plate
 - user-defined plate shape (a closed polyline)

- Insertion plane define the plane of plate insertion as upper, lower, middle, or free. You can also specify an offset value denoting the distance to the insertion plane relative to UCS
- In the middle is a graphical field that displays a defined plate positioned in relation to the insertion point. You can define offsets graphically, by using the cursor to shift the plate with respect to the point or by entering dimension values in the edit fields (not available for user-defined shape).
 - Circular plate radius (R) and offset (Dr) that determines the distance of the insertion point from the center of a circular plate
 - Rectangular plate width and length (W and L) of the sides and offsets (Dx and Dy) that determine distances between the insertion point and center of a rectangular plate

Under the schematic drawing, there is the More >> (or Less <<) button, which opens (closes) an additional field where a surface finishing type for a plate (for example, hot-dip galvanized, electrogalvanized, anticorrosive paint, fireproof coat) can be selected. A default type of surface finishing can be selected in the Project preferences dialog on the Materials tab; this dialog also allows adding a new surface finishing type or deleting a selected type from the list of available surface finishing types.

- Use the icons on the right to define the plate insertion:
 - point
 - 뢱 polyline
 - 🖵 Pick contour –available for user-defined plates
 - Pick internal point available for user-defined plates
 - Match properties used to adopt (inherit) parameters from a defined plate

When the Plate dialog displays, it displays the plate parameters defined most recently.

7.2. Plate definition

To define a plate:

- 1. Select the plate shape type (rectangular, circular, or user-defined).
- 2. Define plate dimensions (number of dimensions depends on selected plate shape).
- 3. Define plate thickness.
- 4. Specify the plate material.
- 5. Determine a family to which a plate will belong.
- 6. If needed, select a surface finishing type (after you click More >>).
- Determine a plane of plate insertion and a desired offset value, denoting the distance to the insertion plane relative to UCS.

After you have defined the plate, specify its method of insertion:



Specify the insertion point and direction

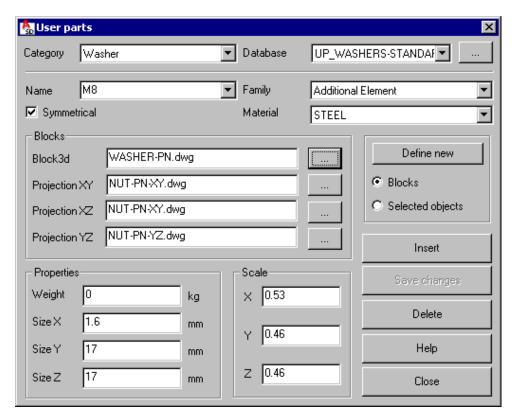
Define a polyline, rectangle, or circle depending on the selected type of plate Select an existing contour (polyline) - available for user-defined plates Specify an internal point of the existing contour (polyline) - available for user-defined plates

8. USER PARTS

8.1. User parts

Use this option to define user parts (non-standard structure elements, such as prefabricated elements, user-defined elements, and so on). User parts have properties of a standard element (weight, position symbol, and so on); these parts are presented in documentation. To begin defining user parts, open the User parts dialog from:

- Menu: Steel / User parts
- Ribbon: ASD Model / Elements / User parts
- Toolbar: General / User parts
 Command line: RBCS_USERPART.



Objects that are user parts are defined in the model layout, and are displayed on printouts. Their display on printouts is determined by blocks linked with user parts. Blocks may consist of any AutoCAD® objects.

User parts are saved to databases which, in turn, are grouped into categories (one category may include any number of databases). Databases are separate XML format files, and each database has a description displayed in the dialog. You can add categories and add databases in selected categories.



A database may belong only to one category.

User parts that belong to one or several databases may be linked with the same blocks. A block must exist and be saved on the hard disk. User parts included in databases are linked with blocks (DWG format files), which determine the way they are displayed (symbol) in a model and on printouts.

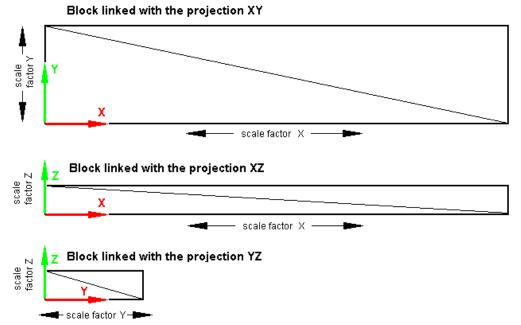
User parts belong to families, which determine their properties such as color, layer, and so on. User parts may be assigned positions, but the software does not check geometry of user parts during positioning. The criteria for identification of user parts are name, category, and properties saved in the database.

A user part weight defined in the User parts dialog is taken into account while calculating the weight of a selected structure part and the center of gravity of a structure.

For a chosen part, the block may be scaled independently in the directions of the local coordinate system of an object. Scale factors refer to the directions determined by the local coordinate system of a 3D block. The same scale factors are applied both to a 3D block in the model layout and to 2D blocks presenting user parts in drawings.

After user parts are positioned, they can be drawn as single parts or be displayed in drawings of assemblies or groups. Drawings of user parts are generated automatically, considering the location and orientation (the local coordinate system) of a part defined in the model layout. In drawings, user parts are shown as blocks linked with them.

The following drawing illustrates which scale factors are used for views and directions.



User parts are displayed in drawings only in orthogonal projections.

User parts shown in drawings of assemblies or groups may be displayed using the line type and the line color chosen in a dimensioning style.

User parts cannot be dimensioned; descriptions for them are generated in the same way as for single parts (with the possibility to use variables).

User parts defined in the model layout are modifiable, but not all parameters of user parts can be changed.

It is possible to execute basic AutoCAD® edit commands (such as copying and translation) on user parts.

User parts may be components of assemblies, groups, or schemes.



NOTE:

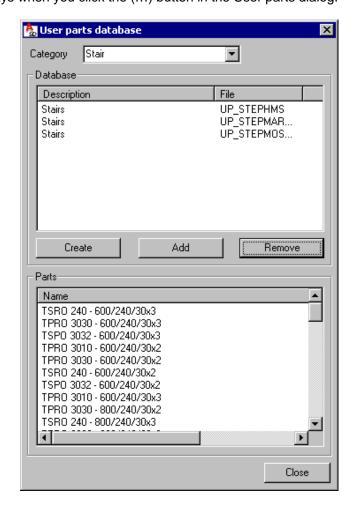
It is not possible to perform the machining operations using the AutoCAD® Structural Detailing options on user parts.

See also: Definition of a new user part User part database

8.2. User part database

Using the User parts database dialog, you can perform operations in databases and categories. It also lets you view the contents of individual databases.

The dialog displays when you click the (...) button in the User parts dialog.



Perform basic operations:

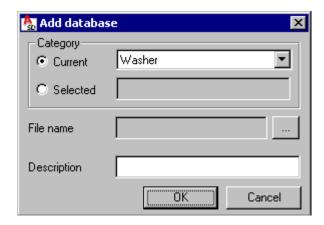
Add a new category



After clicking Add new, you can specify the category name and the category description and file for the new category.

Create a new database

Lets you create a new database (file) in the default folder; the database will belong to a category chosen from the selection list.



In the dialog, specify a name and a description of the database.

8.3. Definition of a new user part

To define a new user part:

Click the (...) button in the top part of the User parts dialog to add a new user part to the current database belonging to a selected category.

A part may be added:

- By blocks using this option, you add an object by linking parts with blocks saved in external DWG files.
 - After you click Define new, specify a part name in the Name field. Values in the remaining edit fields for the newly defined user part are inherited from the previous user part.
 - In the central part of the dialog, specify access paths to external DWG files. The insertion point and the local coordinate system of an object are defined in DWG files. In the lower part of the dialog determine parameters of the user part (a weight, a scale and dimensions for individual directions).
- By selection of parts using this option, you add an object by indicating on the screen objects that will make up blocks linked to the user part.
 - After you click Define new, the User parts dialog closes; specify a name of the user part in the command line. Next, the command line shows commands that enable:
 - Selection of objects that will make up the blocks linked with the 3D projection
 - Selection of objects that will make up the blocks linked with the projections: XY, XZ, and YZ

For all objects, it is necessary to determine the base point and a point positioned on the positive side of the axis. Blocks for projections are saved in DWG files. After you have selected objects for the mentioned projections, the User parts dialog displays. Edit fields in the upper and central parts of the dialog are filled out with names given while defining the projections.

In the lower part of the dialog, specify parameters of the user part (weight, scale, and dimensions for individual directions). By default, values of the weight and dimensions equal 0, whereas scale factor values equal 1.0.

Insertion of a user part

Click Insert to insert the selected user part into the model layout at a point you specify indicated.

Saving of modifications

Click Insert to accept changes made in the existing databases (the button is inactive until changes are made).

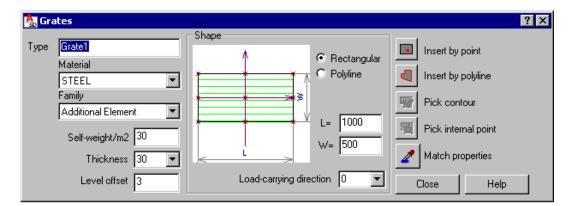
9. GRATES

9.1. Grates

Use this option to define grates.

To begin defining a grate, open the Grates dialog from:

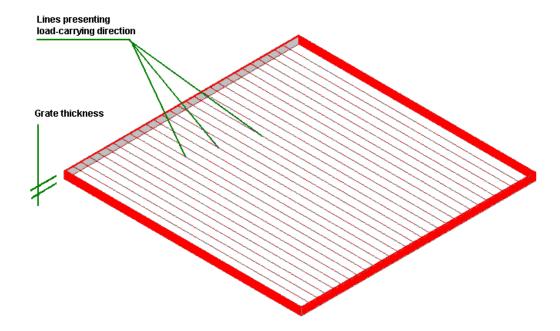
- Menu: Steel > Grates
- Ribbon: ASD Model / Elements / Grates
- Toolbar: General / Grates
 Command line: RBCS GRATE.



The Grates dialog has 3 main parts:

- On the left, you can specify basic information for a defined grate:
 - Type specify a name of the defined grate
 - Material select a material type assigned to the defined grate
 - Family select a family (group) of grates to which the defined grate will be assigned
 - Self-weight based on this value the program calculates a grate weight
 - Thickness select a value of grate thickness or enter a value into the edit field
 - Level offset a value of the distance between the grate insertion point and the bottom left corner of the grate; an offset equals 0, if the bottom left corner of the grate coincides with the grate insertion point
- In the middle is a graphical field that displays the defined grate, and where you can specify a grate shape:
 - Rectangular when this is selected, you can define side dimensions W and L
 - Polyline any shape defined by means of a polyline

You can define a load-carrying direction for the grate (an angle in the XY plane); the direction is displayed in the model layout and drawings using the symbol 1. The longer axis of the symbol is parallel to the load-carrying direction of the grate. The symbol of the load-carrying direction is displayed (in the same color and the same style as the grate edges) only in a top view and a 3D view of the grate in the plane parallel to the grate plane. The grate with its basic elements is illustrated below:



- Use the icon on the right to define the grate insertion:
 - point (not available for a shape defined by polyline)
 - polyline
 - Pick contour available for user-defined grates, but not available for rectangular grates
 - grates
 Pick internal point available for user-defined grates, but not available for rectangular grates
 - Match properties –used to adopt parameters from a defined grate

When the Grates dialog displays, it displays the grate parameters most recently defined.

10. MACHINING

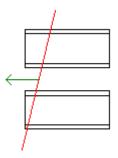
10.1.Fit to line

Use this option to fit a profile to a defined line.

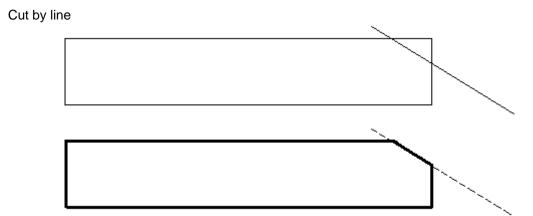
To begin fitting a profile to a line, follow:

- Menu: Steel / Machining / Fit to line
- Ribbon: ASD Model / Machining / Fit to line
- Toolbar: Machining I / Fit to line
- Command line: RBCS_FITTOLINE.

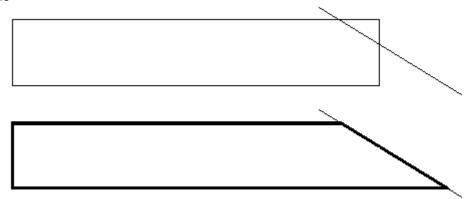
The operation of fitting a profile may be performed using either a line defined using AutoCAD© tools or a line defined by specifying 2 points. You can fit several profiles positioned on the cutting line (see the drawing below).



The Cut by line option operates in a similar manner to Fit to line. The direction is indicated by specifying a point on one side of the cutting line, and the operation determines the part of an object that remains after the cutting operation is performed. The difference between Fit to line and Cut by line is displayed below: the drawings show an object before performing the fitting and cutting operations (together with a defined line), and an object generated after performing the fitting and cutting operations.







10.2. Fit to polyline

Use this option to fit an object to a defined broken line (polyline).

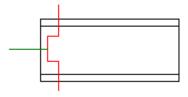
To begin fitting an object to a polyline:

- Menu: Steel / Machining / Fit to polyline
- Ribbon: ASD Model / Machining / Fit to polyline



• Command line: RBCS_FITTOPOLY.

The plane of fitting (cutting off) consists of a few segments, as shown in the drawing below.



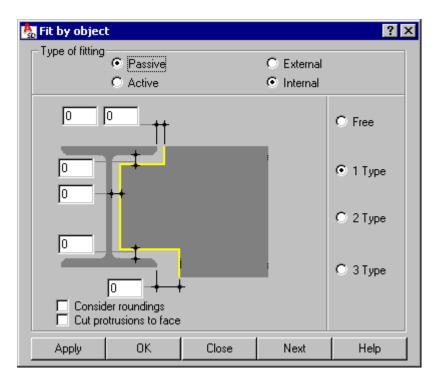
10.3. Fit to object

Use this option to fit 2 structural elements to each other. To begin fitting objects, open the Fit by object dialog from:

- Menu: Steel / Machining / Fit to object
- Ribbon: ASD Model / Machining / Fit to object



In the drawing area, select 2 structure elements that are to be fitted to each other: object that is being adjusted and object to which the first object should be adjusted.



Under Type of fitting, specify the mode of object fitting:

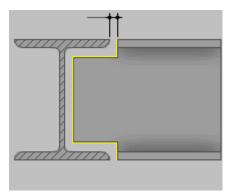
- Passive if this is selected, fitting of objects is not updated after each modification of the fitted object and the object to which it is being fitted.
- Active if this is selected, the software remembers the object to which the selected
 part is being fitted. Fitting of objects is updated after each modification of the fitted
 object and the object to which it is being fitted (after modifying a position of any of the
 objects, the parts will be automatically fitted, applying all the parameters defined in the
 dialog). If the object to which the selected part is being fitted is deleted, fitting of
 objects is deleted, as well.



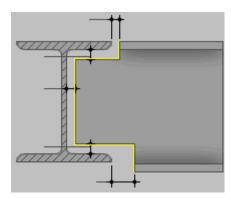
After the fitting of objects operation is completed, the mode of object fitting can be changed in the Properties dialog.

Specify the type of object fitting:

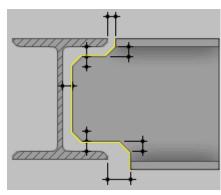
- External one of the selected objects is fitted to the external contour of another object.
- Internal one of the selected objects is fitted to the internal contour of another object. For Internal, there are 4 options:



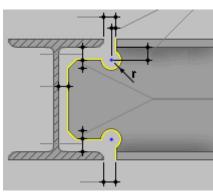
Free - use this method to perform a quick fitting operation. In this case, determining parameter values is not required. If more exact parameter determination is required, apply one of the remaining methods.



1 Type - a distance value may vary for flanges and web of the element undergoing fitting.



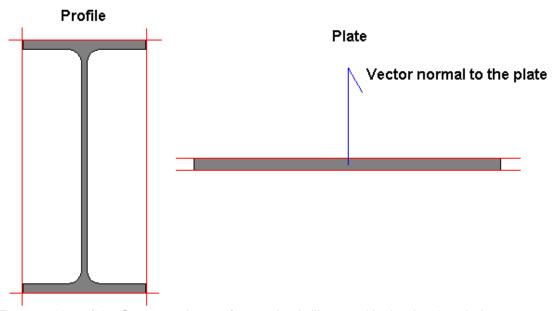
2 Type - for this type of fitting, you can switch on/off chamfering. An angle applied during the chamfering operation always equals 45 degrees.



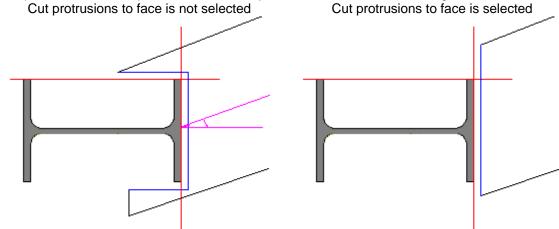
3 Type - using this type of fitting, you can adjust objects with additionally defined openings. Opening parameters include: position of the opening center, translations of the opening center position, and radius value. It is also possible to perform the chamfer operation on some elements of the parts that undergo fitting.

In the bottom part of the dialog are the following options:

- Consider roundings if this is not selected, roundings of rolled profiles will not be considered while generating a cutting plane.
- Cut protrusions to face if this is selected, additional planes that form a contour of a
 part (profile, plate) are accounted for in a cut shape (shown in red in the drawings
 below). Out of these planes, the software selects the one for which the value of the
 inclination angle between the plane and the cut object is closest to 90 degrees.



The operation of the Cut protrusions to face option is illustrated in the drawings below.



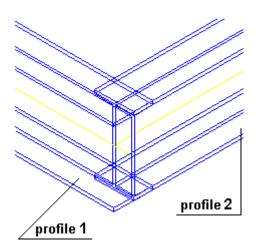
Clicking Apply changes parameters of the selected objects. Click OK to perform the defined operation of object fitting based on the adopted parameters, and close the dialog. Click Close to conduct the defined operation without saving the changes made recently. Click Next to start fitting of successive objects.

See also:

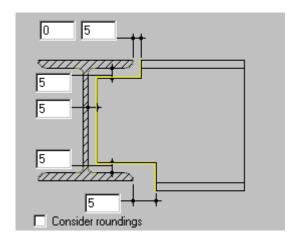
Example of fitting two I-sections to each other

10.4. Example of the fit to object operation

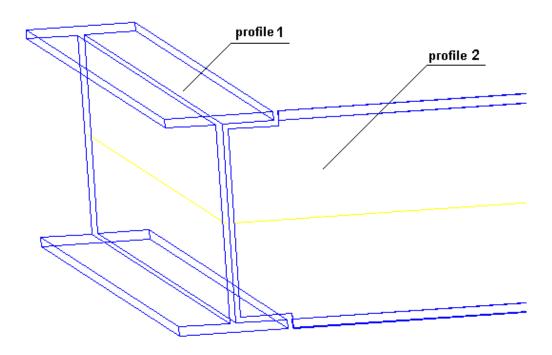
To begin fitting two I-sections, as illustrated in the drawing below:



- Select the Fit to object option
- In the drawing area, select the profile to which the other profile should be fitted (profile 1) and the profile which should be fitted to profile 1 (profile 2).
- In the Fit by object dialog:
 - o For Type of fitting, select Passive and Internal.
 - o For Type, select 1 Type.
 - o Clear Consider roundings.
 - o Specify the fitting dimensions as displayed in the drawing below.



Click Apply.
 The I-sections fitted are displayed in the following drawing.

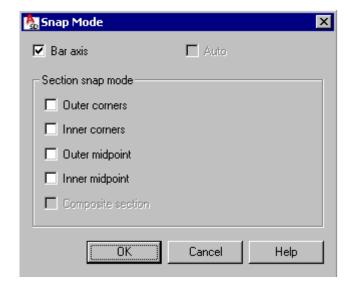


10.5. Object snap settings

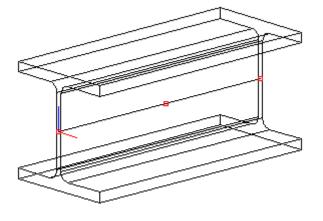
While making a drawing, you can position the cursor at snap points of objects (profiles or plates).

To begin specifying snap settings, open the Snap Mode dialog from:

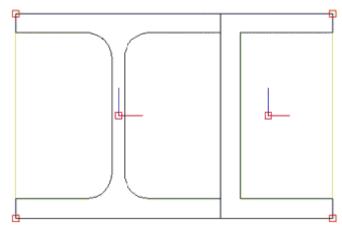
- Menu: Steel / Tools / Workspace / Object snap settings
- Ribbon: ASD Model / Tools / Object snap settings
- Toolbar: Workspace / Object snap settings
- Command line: RBCS_PSNAP.



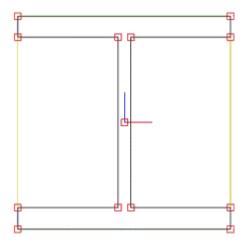
Specify snap points:



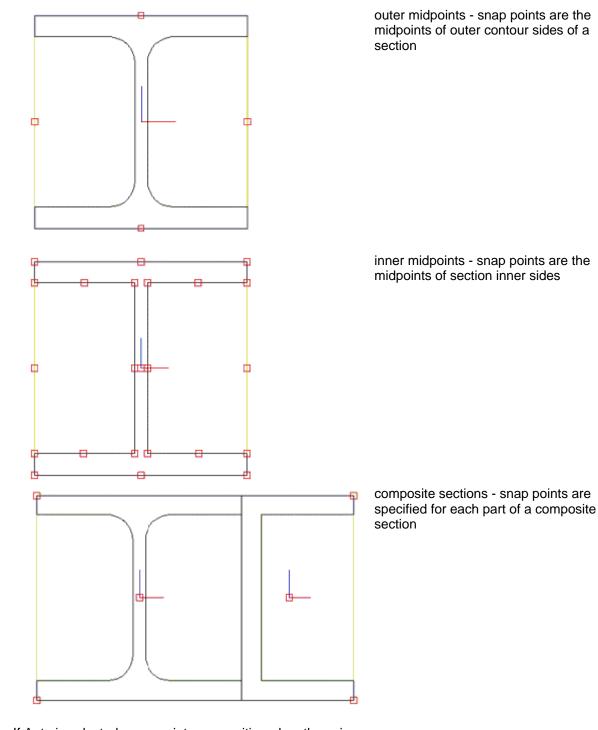
axis - snap point: plate - plate center profile - 3 snap points on the bar axis: beginning, middle, and end



outer corners - snap points are the corners of the outer contour circumscribed on a cross-section



inner corners - snap points are the inner corners of a section as shown in the drawing

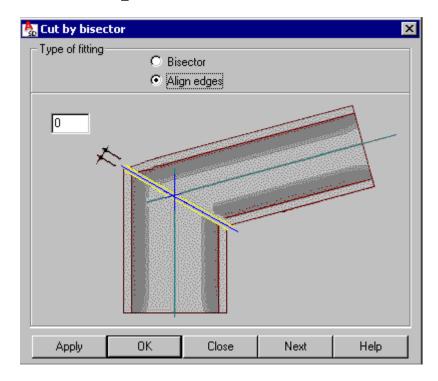


If Auto is selected, snap points are positioned on the axis.

10.6.Cut by bisector

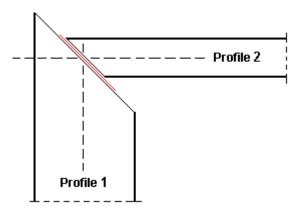
Use this option to fit 2 profiles to each other using a bisector. To begin the fitting operation, open the Cut by bisector dialog from:

- Menu: Steel / Machining / Cut by bisector
- · Ribbon: ASD Model / Machining / Cut by bisector
- Toolbar: Machining / Cut by bisector
- Command line: RBCS_CUTBISEC.

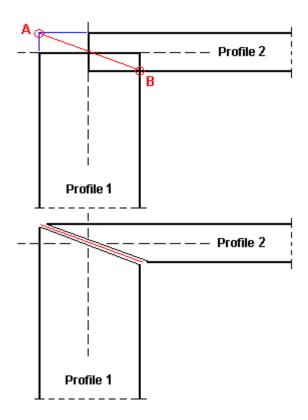


The cutting operation may be performed as cutting with:

A bisector, the result of which is a cut as shown in the drawing below.



• Alignment of profile edges (the cutting line goes from point A to point B), the result of which is a cut as shown in the drawing below.



For the both types of cutting, it is possible to define a distance between the cut profiles.

When you click Apply, parameters in selected objects are modified. Clicking OK performs the defined operation of cutting the profiles based on the adopted parameters and closes the dialog.

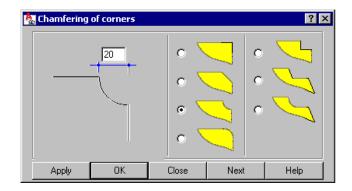
After you click Close, the defined operation is performed without saving the changes made recently. Cutting of successive profiles starts when you click Next.

10.7. Chamfering of corners - plates

Use this option to chamfer plate corners.

To begin chamfering plates, open the Chamfering of corners dialog from:

- Menu: Steel / Machining / Chamfer plate corner
- Ribbon: ASD Model / Machining / Chamfer plate corner
- Toolbar: Chamfer plate corner
- Command line: RBCS_CHAMFCORN
- In the drawing area, select a plate and the plate corner.



After you select the chamfer type, define chamfer dimensions (the number of available dimensions depends on the selected chamfer type). The chamfer is an active object, which means that it may be edited later (change of chamfer type, modification of parameter values). It is also possible to delete a plate chamfer.

Click Apply to modify chamfer parameters. Click OK to run the defined chamfer operation based on the parameters adopted and close the dialog.

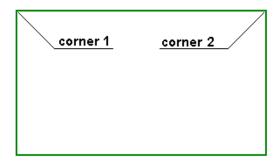
After you click Close, the software carries out the defined operation without saving the changes made recently. Clicking Next starts chamfering of the next plate corner.

See also:

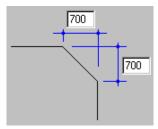
Example of chamfering a plate corner

10.8. Example of chamfering plate corners

To begin chamfering corners of the plate, as shown in the drawing below:



- Click the Chamfer plate corner option
- In the drawing area, select the plate whose corners will be chamfered.
- Select the corners of the plate that should be chamfered (in the drawing above, these are corner 1 and corner 2).
- In the Chamfering of corners dialog:
 - For chamfering type, select
 - Specify the chamfer dimensions as shown in the drawing below.



Click Apply.
 The chamfered plate is illustrated in the drawing below.



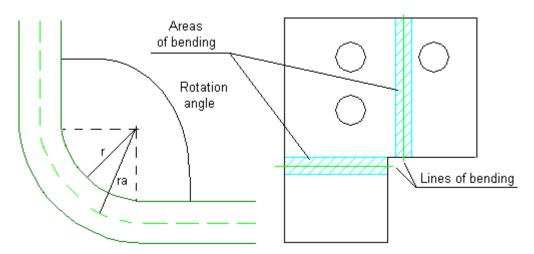
10.9. Bending of plates

A bent plate is an element made by bending a standard plate defined in a model. Bending of plates is a reversible operation.

After you perform the bending operation, a plate becomes a bent plate in the software.

A bend in a plate may be defined by specifying (see the drawing below):

- Bending line
- Angle of bending
- Reference surface this determines whether a radius is defined on the internal surface of the plate (the **r** radius) or on the middle surface of the plate (the **ra** radius)
- Side of the plate to be bent





Areas of bending may not overlap.

While you define a bend in a plate, the following commands will display in the command line:

Choose method [by2Points/byLine]:

Specify start point:

Specify end point:

or

Select line:

Location of bending line [Center/Edge]

Specify internal radius or [Axial]:

Specify angle:

Pick side to be wrapped:

10.10. Insert arc - profiles

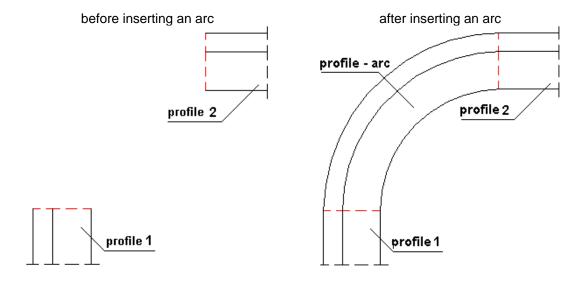
Use this option to connect (by means of an arc element) 2 profiles not touching each other, lying in the same plane.

The option is available from:

- Menu: Steel / Machining / Insert arc
- Ribbon: ASD Model / Machining / Insert arc
- Toolbar: Machining / Insert arc
- Command line: RBCS_BENDCORNER.

To insert an arc between 2 profiles, see the drawing below:

- 1. Click the Insert arc option
- 2. In the drawing area, select 2 profiles to be connected by an arc.
- 3. Specify a radius of the inserted arc.
- 4. Press Enter.



10.11. Copy connection

Using this option, you can copy all types of connections defined in AutoCAD® Structural Detailing. Copying comprises all the elements of a connection.

To define a connection:

- 1. Click the Copy connection option
- 2. In the drawing area, select the connection to be copied.
- 3. Select the profiles to which the selected connection will be ascribed. If all profiles in the structure model are selected, the software will find all the profiles that can be ascribed the selected connection.
- 4. Press Enter.

A connection will be copied if the following conditions (for profiles in the copied connection and profiles where the connection will be copied) are satisfied:

- Target profiles are identical to those in the indicated connection
- Mutual orientation of profiles is identical to that in the indicated connection



If a connection involves machining operations that you have defined manually, they are deleted from the elements of this connection when regenerating this connection.

10.12. Collision detection

Using this option, you can find structure elements that overlap.

To detect a collision of elements in the structure model:



- 1. Click the Collision detection option
- 2. In the drawing area, select elements of the structure model to be verified. If all elements in the structure model are selected, the software will find all the collisions of structure component elements.
- 3. Press Enter.

After you have completed the verification of the structure model, the number of collisions found between structure elements (profiles, plates, bolts, and so on) displays in the command line. Collision sites are marked in the structure model.

It is possible to save collision tests; they are saved in the RBCS_Collision_Detection layer and represented by AutoCAD® 3D Solid objects.

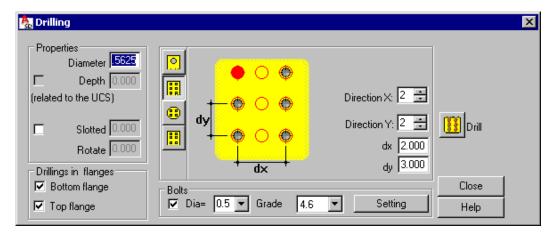
DRILLS/BOLTS/WELDS 11.

11.1.Drills

Use this option to define bolt openings.

To begin defining bolt openings, display the Drilling dialog from:

- Menu: Steel / Machining / Drill/Bolt
- Ribbon: ASD Model / Bolts/Welds / Drill/Bolt
- Toolbar: Machining II / Drill/Bolt
- Command line: RBCS DRILL.



Specify parameters:

- Under Properties, define basic information for bolt openings After you select Depth, an edit field becomes accessible in which you can define the drilling depth. If Depth is not selected, an opening is drilled through a whole element. After you select Slotted, the defined opening will be slot-shaped, and you can specify length of the opening rectilinear part and opening rotation angle, in respective edit fields. If Slotted is not selected, the opening will be circular. Under Drillings in flanges, you can select Bottom flange or Top flange in order to
 - position drillings in the bottom/top flange of a member section. In the middle part of the dialog, select a mode of defining bolt openings / drillings (the

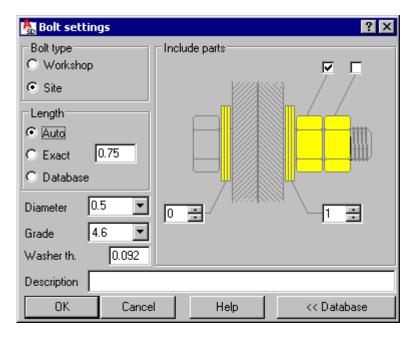
selected mode is displayed in the preview area)

- or single bolt opening
- rectangular arrangement of openings / drillings with a regular spacing of openings in both directions. When this opening type is selected, the following parameters need to be defined:
- o Number of rows and columns of openings / drillings (direction X and direction Y)
- o Distances between rows and columns of openings / drillings (dx and dy)
- polar arrangement of openings / drillings (openings will be defined on an arc of a circle). When this opening type is selected, the following parameters need to be defined:
- Number of openings / drillings
- o Radius of a circle on which the centers of openings / drillings will be positioned
- Angle between the successive openings/bolts. If Auto fill is selected, openings will be arranged along the entire circumference length and the angle will be determined automatically.
- rectangular arrangement of openings / drillings with an irregular spacing of openings. When this opening type is selected, the following parameters need to be defined:
- Number of rows and columns of openings / drillings (direction X and direction Y)
- Distances between successive openings in rows and columns of openings / drillings (dx and dy) (for example, 50 75 50 100)
- If Bolts is not selected, bolts will not be placed in the openings (all the edit fields are
 inaccessible). If the option is selected, bolts will be placed in the openings and the
 fields for defining bolt parameters (Diameter and Grade), are accessible. Click Setting
 to open the Bolt Settings dialog.
- (Drill) click this in order to select objects in which the openings are to be made and drill bolt openings based on parameters defined in the dialog.

The dialog opens showing the opening/bolt parameters defined recently.

11.2.Bolt settings

The Bolt settings dialog displays after you click Setting in the Drilling dialog.



On the left, specify basic information for applied bolts:

- Bolt type (Workshop or Site)
- Washer thickness in AutoCAD® Structural Detailing, bolts are generated with washers, and the washer thickness is taken into account during calculation of bolt length
- Length
 - Auto determined automatically
 - Exact user-specified
 - Database selected or calculated based on data in a chosen bolt database (see: Principles of calculation of the bolt length)
- Diameter
- Grade
- Description

If you select an option under Include parts, in the selected part is added:

- Nuts (1 or 2)
- Washers head washers or nut washers

When you click Database, the bottom panel of the Bolt settings dialog displays. It contains a table with data regarding bolts, nuts, and washers available in the selected bolt folder.

11.3. Principles of calculation of the bolt length

If you select Database in the Bolt settings dialog, a bolt of the appropriate length is automatically selected from an available bolt database.

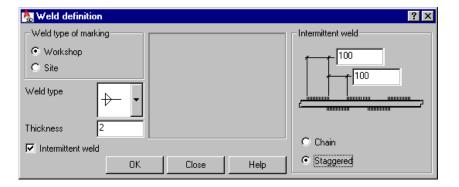
Data needed to determine a bolt length:

- · Sum of thicknesses of all the elements connected with each other
- Bolt parameters:
 - Diameter
 - Type connected with a selected bolt database
- Washer parameters:
 - Number
 - Type connected with a selected washer database
- Nut parameters:
 - Number
 - o Type connected with a selected nut database

The bolt list displayed in the lower part of the Bolt settings dialog includes all bolts of a given diameter from a database. The bolt found is highlighted on this list.

11.4. Weld definition

Use the Weld definition dialog to define weld parameters.



After you have selected the weld type of marking (workshop or site), you can determine parameters of the defined weld:

- Weld type
- Weld thickness (calculated automatically)
- Dimension(s) for edge preparation are shown in a schematic drawing the number of parameters depends on a weld type

If you select Intermittent weld, you can define an intermittent fillet weld (composed of many segments of single welds). The drawings below include the following designations:

PE - beginning of a weld

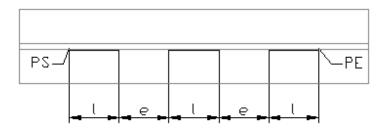
PS - end of a weld

I - length of weld segments

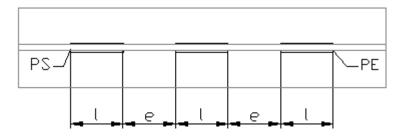
e - spacing between weld segments

All segments of a weld are of the same length (I); spacings between segments of welds are also of the same length.

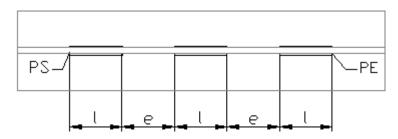
The following types of intermittent welds are available: Intermittent weld - one-sided



Intermittent weld - double-sided - chain



Intermittent weld - double-sided - staggered



See also:

Weld description styles

12. POSITIONING

12.1.Position - definition and description

The design process in AutoCAD® Structural Detailing may be divided into 2 primary stages: model definition and preparation of drawing documentation. The intermediate stage involves positioning (assigning) to structure elements the position designations, which are used to identify elements in the documentation. Assigning a position is necessary in order to prepare a drawing of a given element.

An additional objective of positioning is to limit a number of drawings in such a manner that identical structure parts are displayed in a single drawing. There are 3 types of objects that may be positioned in AutoCAD® Structural Detailing:

- Single parts
- Assemblies (only automatic positioning)
- Groups

The position is displayed in the form of an element (element group) label. The position syntax consists of a Prefix + number, where:

- Prefix is short, editable text (for example, pos) that may be linked to the element's family (single part, groups, and so on.). This part of a name may be disregarded.
- Number is saved in the numeric or alphanumeric format (it may be ascribed automatically or manually). As a result of adopting a position number, the classification is set properly on the list.

Positions may be assigned independently on 2 positioning levels (positioning is carried out separately for each of the levels listed):

- Single part
- Assemblies

The program allows 2 methods of assigning a position:

- Manual positioning manual definition or modification of positions for selected elements single parts or groups.
- Automatic positioning automatic definition of a position for the entire model or its selected part.

Each of the positioning levels listed is connected with the type of drawings (the template used for automatic drawing generation): single part drawing, workshop drawing, and site drawing. Elements that may be chosen for individual positioning levels:

Single part Single parts: profiles, plates, connectors

Assembly Single parts, groups

Groups Groups

To access the Remove position option, go to:

• Menu: Steel / Positions / Remove position

• Ribbon: ASD - Model / Positioning / Remove position

Toolbar: General / Remove position

Command line: RBCS_REMPOS.

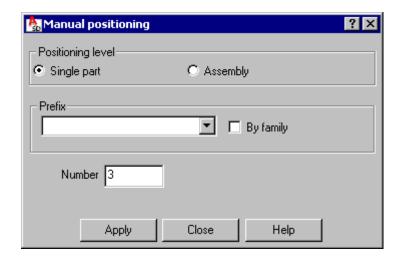
To remove the positions assigned to elements, select the elements and activate the Remove position option. Positions will be removed from all selected elements.

12.2. Manual positioning

Use this option to assign a position manually.

To begin assigning positions manually, open the Manual positioning dialog from:

- Menu: Steel / Positions / Assign position
- Ribbon: ASD Model / Positioning / Assign position
- Toolbar: General / Assign position
- Command line: RBCS MANUALPOS.



To assign a position manually:

- 1. Select an element or several identical elements
- 2. In the Manual positioning dialog, specify:
 - Positioning level
 - Position label (prefix, number, and comment see the position description)
- 3. Click Apply.



When you assign a position manually, only basic criteria of element recognition are considered. User-defined criteria are available only during automatic positioning.

You can also assign (modify) a position at any time while working in AutoCAD® Structural Detailing by means of the Object properties option on the Inspector dialog.

See also:

Position verification

12.3. Automatic positioning

Use this option to assign positions automatically.

To begin assigning positions, open the Automatic positioning dialog from:

- Menu: Steel / Positions / Run Automatic positioning
- Ribbon: ASD Model / Positioning / Automatic positioning
- Toolbar: General / Automatic positioning
- Command line: RBCS_AUTOPOS.

You can assign positions automatically for a whole structure model or for its selected part.

After you select structure elements, use the tabs on the Automatic positioning dialog to specify parameters:

- General
- Additional
- Numbering.

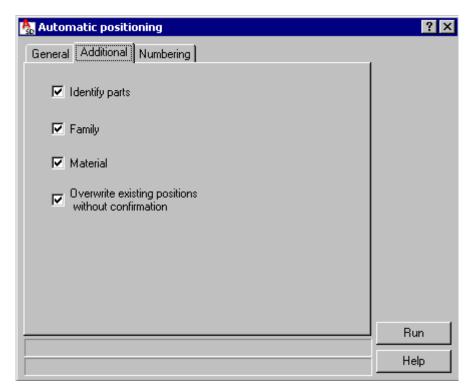
Click Run to begin the automatic positioning operation for all selected structure elements. All positions assigned during the positioning operation are given an identical prefix and comment.

See also:

- Position verification
- Example of automatic positioning

12.4. Additional

On the Additional tab of the Automatic positioning dialog, you can specify parameters to be considered during the positioning operation.



Available options:

- Identify parts
 - If you select this (it is selected by default), the software checks the identity of a structure part. If 2 identical parts are found, they are assigned the same position. If you clear this option, each structure part is assigned a successive position.
- Family
 If you select this, the software considers families while checking the identity of a structure part. If structure elements belong to different families, these structure elements will be ascribed different positions.
- Material
 If you select this, the software considers materials while checking the identity of a structure part. If structure elements are assigned different materials, these structure elements will be ascribed different positions.
- Overwrite existing positions without confirmation

If you select this, the structure elements that have been selected and assigned positions are automatically ascribed new positions (previous positions are overwritten).

If you clear this option and the software finds an element with a position already assigned, the command line prompts you about overwriting the position:

No - the position will not be changed

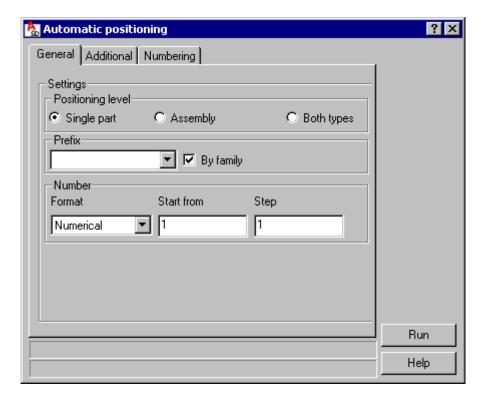
Yes - the position will be changed to a new one

Yes for all - the position will be changed for all the successive elements that have already been assigned positions.

If the Identify parts option is selected and several identical parts have been found during positioning, the number ascribed to the element is the number of the first identical part found during positioning. All the parts identical to a previously positioned element are ascribed the same position as the first element found.

12.5.General

On the General tab of the Automatic positioning dialog, you can specify parameters for level, prefix, and number.



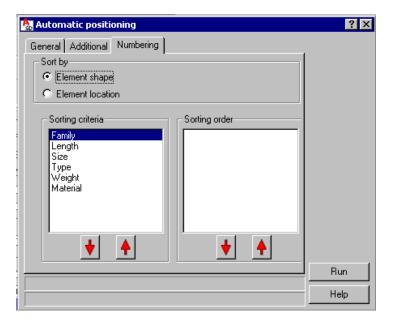
Available options under Settings:

- Positioning level the selected positioning level will be assigned to all elements chosen for the positioning operation
 Single part - groups are not considered in the positioning process
 Assembly - positions are assigned both to single elements and to groups
 Both types - single parts are not considered in the positioning process
- Prefix from the list, you can select a character string that will be ascribed as a prefix to all positions. You can also select By family to take a prefix from a family
- Number specify the format of position numbering
 - Format Alpha or Numerical
 - Start from specify the beginning value for the position numbering; successive positions will be numbered based on the value specified for Step When the entire alphabet has been used, position numbering will use the following form: AA, AB, AC, and so on.
 - Step define the increment value for position numbering.

12.6. Numbering

Use the Numbering tab on the Automatic positioning dialog to specify sorting criteria for position numbering. The available options depend on the selection for Sort by.

Sort by element shape

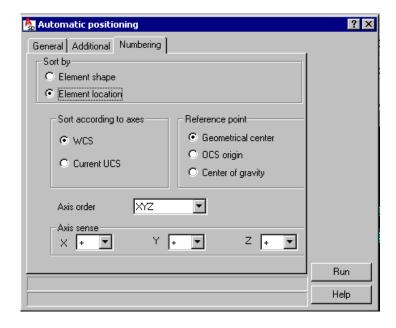


In this mode, the numbering order is determined based on selected properties of the element shape.

Specify additional sorting parameters:

- Sorting criteria (Family, Length, Size, Type, Weight, and Material) highlight a
 parameter and click the ↑↓ buttons to move the selected parameter up or down the
 list, thus defining the required sorting order.
- Sorting order (ascending) of quantities defined in the software for a selected sorting
 parameter (for example, all profile types) highlight a parameter and click ↑↓ buttons
 to move the selected property up or down the list, thus defining the required sorting
 order.

Sort by element location



In this mode, successive numbers are ascribed based on the position of the element reference point in the coordinate system.

Specify additional sorting parameters:

- Sort according to axes choose whether the location of positioned elements is to be verified with respect to the global coordinate system (WCS) or with respect to the current user local coordinate system (UCS).
- Reference point determine the point that identifies the point position in space.
- Axes order select the method of searching the parts in the selected coordinate system.
- Axis sense determine if the numbering increases according to the axis sense (+) or if the numbering increases in the direction opposite to the axis sense (-).

12.7.Position verification

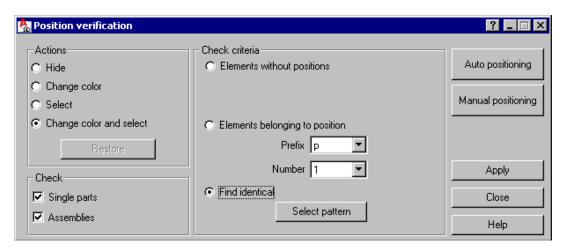
Use this option to control designations of positions ascribed to elements in the process of manual or automatic positioning.

Positioning is verified through searching for positions that either fulfill or do not fulfill selected conditions of positioning.

To begin verifying positioning for the entire structure model, open the Positions verification dialog from:

- Menu: Steel / Positions / Verify positions
- Ribbon: ASD Model / Positioning / Verify positions
- Toolbar: General / Verify positions
- Command line: RBCS_VERIFYPOS

Verification of positions is performed for the entire structure model.



Under Actions, select an operation to perform on elements found during position verification (Hide, Change color, Select, or Change color and select). The effect of an action is remembered until you click Restore or until the dialog is closed and regeneration is carried out.

Under Check, you can choose elements (single parts or assemblies) to be taken into account during structure verification.

Under Check criteria, indicate the elements to be searched in the process of verification:

- Elements without positions select this in order to search for all elements to which no position has been assigned.
- Elements belonging to position select this in order to search for elements that contain the specified prefix and position number.

 Find identical - select this in order to search for elements that conform with the pattern object you select (group or single part). You can select the pattern object on the screen after you click Select pattern.

Click Apply in order to run the verification (the options in the Check criteria field). After verification is carried out on all objects that satisfy the selected criterion, the operation specified under Actions will be performed. In addition, the command line provides information about the number of elements found that fulfill the defined criterion:

Position verification

Identical elements: 10 found.

If you have specified Select for Actions, you can use the buttons at the top right of the dialog to run positioning of elements found and selected during verification:

- Auto positioning click this to perform automatic positioning for elements found in the verification process.
- Manual positioning click this to perform manual positioning for elements found in the verification process.

12.8. Example of positioning and automatic printout

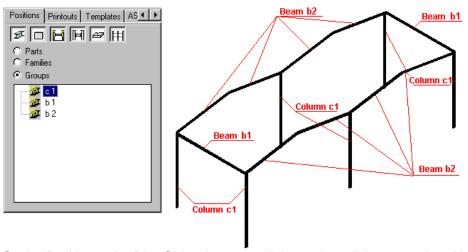
Positioning will be illustrated using a simple 3D frame generated with the Multi-span frame macro available in AutoCAD® Structural Detailing - Steel. To define a multi-span frame:

- 1. Click the Multispan frame option.
- 2. In the drawing area, click to specify the frame insertion point.
- 3. On the Geometry tab of the Parametric Structures 3D Frame Multispan hall dialog:
 - For Number of spans, enter 2.
 - For Number of frames, enter 2.
 - For Frame spacing, enter 8000 mm.
 - Select Rafters and External spandrel beams.
- 4. Accept any user-defined dimensions of the frame that display in the dialog.
- 5. Click the Profiles tab, and specify the attributes for component elements of the frame: For Columns select profile C 10x15.3 and select family Column.
 - For Rafters select profile C 10x15.3 and select family Beam.
 - For Spandrel beams select profile C 10x15.3 and select family Beam.
- 6. Click OK. The software will generate a model of the 3D frame.

The profiles generated in the 3D frame have been added on the Model tab of the Object inspector dialog. To assign positions to the elements of the created frame:

- 1. On the Model tab of the Object inspector dialog, select all the profiles, right-click, and click Auto positioning.
- 2. In the Automatic positioning dialog:
 - On the General tab, for Positioning level, select single part.
 - For Prefix, select By family.
 - Under Number, for Format, select Numerical, for Start from, enter 1, and for Step, enter 1.
 - On the Additional tab, select all the options.
 - On the Numbering tab, for Sort by, select Element shape.
 - For Sorting criteria, select Family.
 - For Sorting order, select Column and Beam.
- 3. Click Run.

All elements of the 3D frame will be assigned positions according to the parameters specified in the Automatic positioning dialog. Because some frame elements are identical, 3 positions will be created on the Positions tab of the Object inspector dialog: columns c1, beams b1, and beams b2. The drawing below illustrates the created positions and location of the structure elements that make up individual positions.



- 4. On the Positions tab of the Object inspector dialog, select all the created positions, right-click, and click Automatic Drawings.
- 5. On the Automatic drawing generation dialog:
 - On the Templates tab, for Column, select Profile 1:10.
 - For Beam, select Profile 1:10.
 - On the Formats and scales tab, for Part type, select single profiles
 - Select Automatic scale.
 - For Format, select A4 ASD.
 - For Arrange views, accept the default values.
 - On the Options tab, for Names of printouts, select Names consistent with position name.
 - For Start printout numbers, select 1.
 - Clear all remaining options.
 - On the Bill of materials tab, select Add table.
 - For Description, select Standard.
 - Select Top Left.

6. Click Generate.

Detailed drawings will be generated for all positions (b1, b2, and c1). The software will create additional layouts (b1, b2, and c1), which will display drawings of selected positions. Also, on the Positions tab of the Object Inspector dialog, documents (projections of positions: views and sections) for each position will be added.

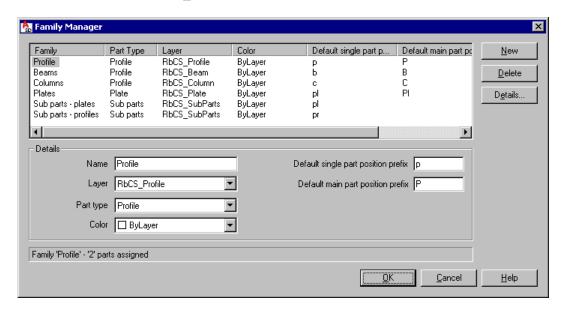
13. FAMILY MANAGER

13.1. Family manager

Use this option to add (remove) families to a project, and to modify family properties.

To begin working with families, open the Family Manager dialog from:

- Menu: Steel / Tools / Family Manager
- Ribbon: ASD Model / Tools / Family Manager
- Toolbar: General / Family Manager
- Command line: RBCS FAMILY.



Basic rules that apply while defining/modifying families in AutoCAD® Structural Detailing:

- The family is an attribute that may be assigned to a profile, bent profile, plate, or user part defined in a structure model. Thus, it is a tool that facilitates management of structure model parts.
- The main goal of defining families is to collect structure elements into certain sets and put them in order so that they are recognizable during filtering, positioning, and so on.
- Any number of families may be defined in a project; a set of families may be saved in a project template (a DWT file).
- The family may be assigned to each of structure parts (profile, bent profile, plate, or user part) during definition of the structure element (dialogs contain the lists that enable selection of a family) or may be applied as an element property after defining the structure elements listed.



The family may be assigned only to a single part (for groups or assemblies, the family is assigned to the main part).

In the Family Manager dialog, define family parameters:

- Name specify the name that identifies a family; a family name cannot repeat in a project.
- Layer this is the default layer on which family elements will be positioned during definition. The connection of the family and layer is not permanent: after definition, an element may be transferred to another layer without losing the family attribute.

- Part Type specify the type of structure element for which the group will be defined.
 The family may be defined for profiles (with bent profiles included), plates, or subparts,
 but a family defined for profiles cannot be assigned to plates (and vice versa).
 Subparts created automatically during operations such as a connection definition are
 ascribed to a default family (based on the option selected in Preferences).
- Color specify the color that will be assigned to parts in the defined family.
- Default single part position prefix enter a prefix (text) to be used during automatic positioning of single parts of the defined family.
- Default main part position prefix enter a prefix (text) to be used during automatic positioning of main parts of the defined family.

Defined families are available in:

- the appropriate selection lists provided in the Profile, Bent profile and Plate dialogs.
- profile or plate properties (families may be modified in the Properties or Inspector dialogs).

Use the buttons at the right of the dialog to work further with families:

- New click this to add a new family to the list of available families. An added family is
 assigned a default name, which you can change. All properties for the new family are
 inherited from the family highlighted on the list. If none of the families has been
 highlighted, the properties are inherited from the first family on the list.
- Delete click this to remove a selected family from the list. All structure elements that were ascribed in the removed family remain in a structure model without the family assigned.
- Details click this to show (or hide) the bottom part of the dialog displayed under the
 list of available families. The options available under Details enable defining
 parameters for a selected family or families. When several families are selected, the
 properties are identical for all selected families.

See also:

Example of definition of a profile family

13.2. Example of definition of a profile family

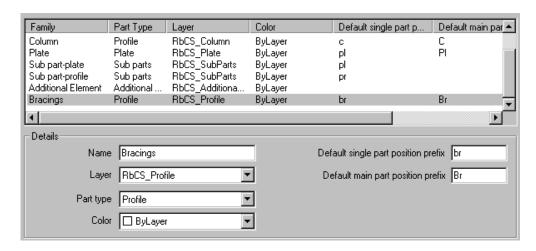
To define a new profile family:



- Click the Family Manager option
- In the Family Manager dialog, click New.
 The software creates a new family with the properties of the most recently selected (highlighted) family.
- Under Details:
 - o For Name, enter Bracings.
 - o For Layer, select RbCS_Profile.
 - o For Part type, select Profile.
 - o For Color, select ByLayer.
 - o For Default single part position prefix, enter br.
 - For Default main part position prefix, enter Br.

The new profile family has been added to the list (see the drawing below).

Click OK.



In the Family Manager dialog, you can also edit parameters of a selected family.

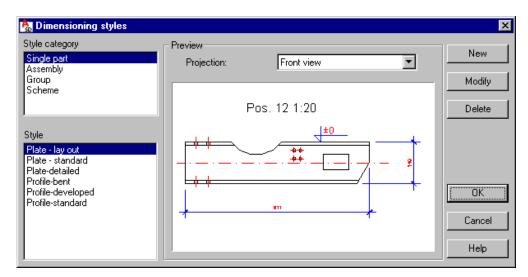
14. DIMENSIONING STYLES

14.1. Dimensioning styles

Use this option to define styles of element dimensioning.

To begin defining styles, open the Dimensioning styles dialog from:

- Menu: Steel / Styles / Dimensioning styles
- Ribbon: ASD Drawings / Styles / Dimensioning styles
- Command line: RBCS_DIM.



Dimensioning consists of positioning elements in a 2D projection of a position:

- Geometrical dimensions: linear dimensions of individual elements or elements joined together to create an element chain, arc dimensions, angle dimensions, diameters, and radial dimensions
- Symbols and names: weld symbols, bolt (opening) symbols, elevation marks, designations of assemblies and groups (profile types, positions), and so on
- Comments (additional text): variables describing created views and presented objects (name, scale, and so on), user-defined text

Dimensioning is performed automatically based on user-defined settings; it is also possible to modify created dimension lines and their descriptions. Dimensioning may be conducted within the edition contour. All dimensional elements (such as dimension lines, text style, and ends of dimension lines) are set in the dimension styles available in AutoCAD®.

You can define any number of dimensioning styles. Each defined style belongs to one of the categories concerning the type of a drawing position (single part, assembly, group, scheme – a group of elements constituting a structure model).

The dimensioning style defines:

- Dimension style from AutoCAD® to be applied
- Elements and the element projections in which the individual elements should be included
- Manner of arranging elements in the drawing

A set of defined styles is saved in a DWG format file or in a template (a DWT file). In the Dimensioning styles dialog, when a drawing position type is selected for Style, all defined styles for the selected category are displayed. A preview of a selected style is shown in the Preview field. For each style, there are different settings defined for each of available element views (top view, front view, side view, 3D view).

One of the styles defined for each drawing position type is a default style, and will be suggested as a default as you define a drawing template. The default type may be changed by highlighting a style within a given category and clicking Default.

Use the buttons on the right of the dialog:

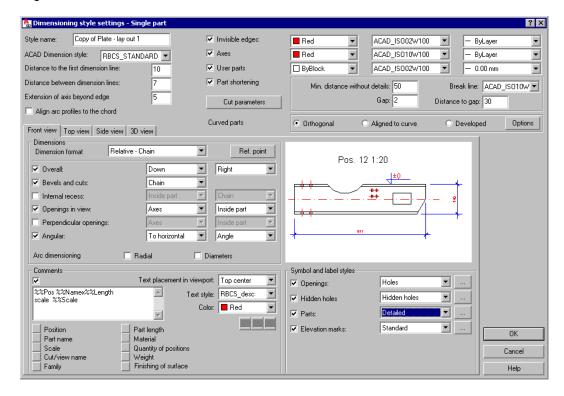
- New click this to open the Dimensioning style settings dialog, where you can define a
 new style for a selected category. The name of a new style and all its settings are
 inherited from the style currently selected.
- Modify click this to open the Dimensioning style settings dialog, where you can change settings of a selected style.
- Delete click this to delete the style currently selected from the list of available styles for a given category.

See also:

Orientation of element views and dimensions in a drawing

14.2. Definition/modification of a dimensioning style

Use the Dimensioning style settings dialog to define a new style or modify an existing dimensioning style. The dialog opens after you click New or Modify in the Dimensioning styles dialog.



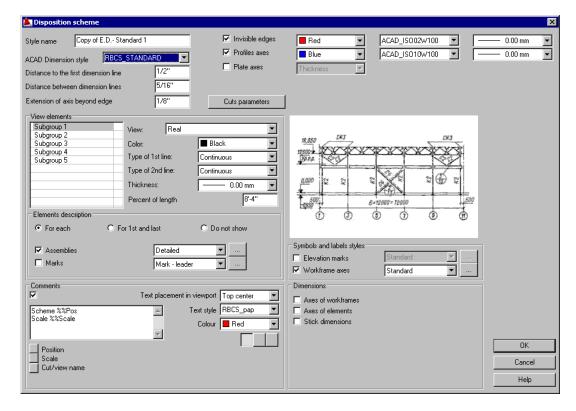
The contents of the dialog depend on the category selected in the Dimensioning styles dialog; there are considerable differences in the part concerned with geometrical dimensions. The dialog also differs considerably if a dimensioning style of disposition scheme has been chosen.

The dialog is divided into the following parts (beginning at the top of the dialog):

- general parameters defined in this part (including a style name) are used in all generated views
- parameters of dimensioned curved parts
- section to define geometrical elements that are to be dimensioned if you select an option on any of the tabs (Front view, Top view, Side view, 3D view), an element selected in the drawing will undergo dimensioning. In the selection fields, you can determine dimensioning parameters.
- a preview of the style currently defined
- a comments section where you can specify additional texts generated automatically for views
- a place to define description styles and symbols you can indicate a symbol that is to be created automatically in the selected style; clicking one of the (...) buttons opens the dialog used for defining description styles ,which determine the manner of displaying a description/symbol in a drawing.

14.3.Definition/modification of a dimensioning styledisposition scheme

Use the Disposition scheme dialog to define a new style or to modify an existing dimensioning style for the disposition scheme (a group of elements constituting a structure model). This dialog opens after you click New or Modify in the Dimensioning styles dialog (the scheme has to be selected as a style category).



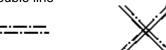
Define parameters using the options at the top of the dialog:

- ACAD Dimension style select an AutoCAD® dimensioning style that will be applied
 in the dimensioning style currently selected. The list contains all the dimensioning
 styles defined in a DWG format file
- Distance to the first dimension line specify the distance between the edge of a dimensioned part and the first dimension line generated automatically
- Distance between dimension lines specify the distance between successive dimension lines
- Extension of axis beyond edge specify the length of lines that extend outside the element edges on both sides of the element
- Invisible edges if this is not selected, invisible edges of an element are hidden in a final drawing. If this is selected, invisible edges are displayed in a drawing (specify their line color, line type, and line thickness parameters at the right of the option)
- Profile axes if this is not selected, profile axes are hidden in a final drawing. If this is selected, profile axes are displayed in a drawing (specify their line color, line type, and line thickness parameters at the right of the option). Profile axes are displayed in all projections; in final drawings, axes are extended by 5 mm outside the edge of a dimensioned object

Click the **Cuts parameters** button in the upper part of the dialog to open the Cut parameters dialog where you can specify parameters of hatching the cut and parameters for designating the cut in disposition schemes.

Specify view parameters (color, type of the first and/or second line, line thickness) using the options in the center of the dialog. Select a view type:

- Real a real view of each element
- Rectangular box a view of each element is represented as a rectangular contour
- Line a view of each element is represented as a single line
- Double line



Column line





Truss scheme



- Rectangular box group for all elements of a group (also for elements in subgroups), the view is represented as a rectangular contour circumscribed on elements of the group
- Line group represented as the largest central (single) line of the rectangular contour circumscribed on elements of the group
- Double line group represented as the largest central (double) line of the rectangular contour circumscribed on elements of the group
- Column line group represented as the largest central line (column line) of the rectangular contour circumscribed on elements of the group

Specify when to show designations (for assemblies and marks):

- For each element of a scheme
- Only for the first and last elements
- Do not show

Moreover, you can select styles for assemblies and marks.

You can specify a value of shortening (in percent) of an element length.

Additional parameters:

- Stick dimensions distances of elements from axes in a scheme (if an element lies on the axis, then a 0 dimension is not shown)
- Level marks level symbols, separately for each subgroup of the scheme
- Axial dimensions dimensions from axis to axis of elements of a scheme
- Level dimensions additional dimension lines of elements (on the side of a scheme, near the level marks); a dimension chain from the lowest to the highest level in the scheme is created.

If Comments is selected, a title (name) is added to the views of a dimensioned object. The title may be text (comprising one or several lines) containing variables or any user-defined character string. Below the text field are variables that you can add to the defined comment at the cursor position. The variable text is displayed with the comment in the edit field.

Available comment variables:

%%Pos - if this is selected, description of a position is added to the text %%Scale - if this is selected, a scale (for example, 1:20) is added to the text %%Vname - if this is selected, an automatic name of a view or section is added to the text

You can determine the placement and appearance of a comment in a drawing:

- Text placement in viewport
- Text style
- Alignment (align right, align left, center)

Under Symbol and layer styles, select styles for the following descriptions and symbols (designations) presented in final drawings:

- Elevation marks
- Workframe axes

14.4.Orientation of element views and dimensions in a drawing

To define position projections, the following terms have been adopted:

- Front view XZ plane of the element local coordinate system
- Top view XY plane of the element local coordinate system
- Side view YZ plane of the element local coordinate system
- 3D view isometric view

The element local coordinate system for a selected position type is defined as follows: Single parts

- Profiles:
- X axis along the profile axis
- Y axis section horizontal axes identically as in the case of profile saving in the database (for most profiles it is the axis of a larger moment of inertia)
- Z axis section vertical axes identically as the case of profile saving in the database (for most profiles it is the axis of a smaller moment of inertia)

- Plates:
- Rectangular

X axis - along the plate length

Y axis - along the plate width

Z axis - perpendicularly to the plate plane

o Circular

X axis - along the radius (diameter) determined during plate definition

Y axis - perpendicularly to X axis (in the plate plane)

Z axis - perpendicularly to the plate plane

User-defined

X axis - along the longest segment belonging to the plate contour

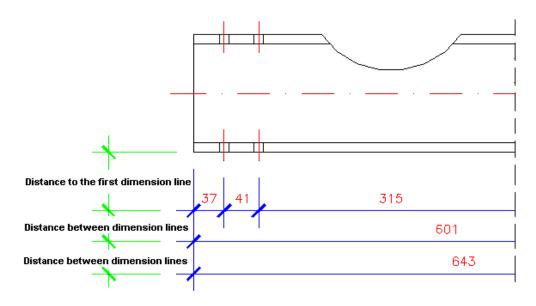
Y axis - perpendicularly to X axis (in the plate plane)

Z axis - perpendicularly to the plate plane

14.5. General part

At the top of the dialog, specify general parameters:

- ACAD dimension style select an AutoCAD® dimensioning style that will be applied in the dimensioning style currently selected. The list contains all the dimensioning styles defined in a DWG format file
- Distance to the first dimension line specify the distance between the edge of a dimensioned part and the first dimension line generated automatically
- Distance between dimension lines specify the distance between successive dimension lines (see drawing below)
- Extension of axis beyond edge specify the length that lines extend outside the element edges on both sides of the element

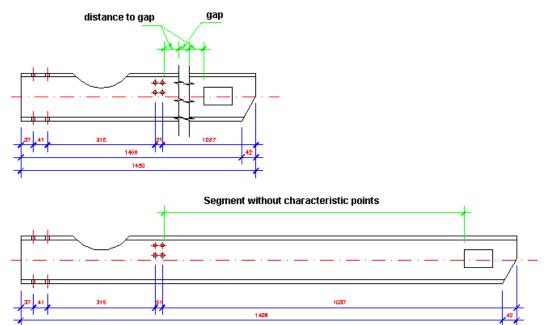


Parts shortening - if this is selected, long elements (element segments that do not
include any characteristic points) will be shortened in a final drawing. The element
shortening is displayed as a gap in a shortened element.

Element shortening will be performed if the element segment (not including any characteristic points) is longer than a value of the user-defined minimal distance without details. The drawing below illustrates the same element, shortened (top part of the drawing) and not shortened (bottom part of the drawing).

You can define a type of break line to use during element shortening; the list contains all available line types in AutoCAD®, saved in a DWG format file. Thickness and color of the break line are identical to those of an element contour.

Additional parameters of shortening can be specified (see drawing below): gap value, and value of the distance to gap (distance between a characteristic point and a gap).



- Invisible edges if this is not selected, invisible edges of an element are hidden in a
 final drawing. If this is selected, invisible edges are displayed in a drawing (specify
 their line color, line type, and line thickness parameters at the right of the option)
- Axes if this is not selected, profile axes are hidden in a final drawing. If this is selected, profile axes are displayed in a drawing (specify their line color, line type, and line thickness parameters at the right of the option). Profile axes are displayed in all projections; in final drawings, axes are extended by 5 mm outside an edge of a dimensioned object
- User parts if this is not selected, user parts are hidden in a final drawing. If this is selected, user parts are displayed in a drawing (specify their line color, line type, and line thickness parameters at the right of the option)
- Cut parameters click this to open another dialog, in which you can specify parameters of section hatching and parameters of designation of a part section.

For styles of assemblies or groups, Distance to the first dimension line indicates the distance measured to the edge determined by a rectangle describing an assembly projection (rectangle sides are parallel to the axes of the element local coordinate system).

When defining a modification of an assembly style, the Draw parts in actual location option is available. If this is not selected, projections of an assembly are generated in the coordinate system determined by the main part of the assembly. If this is selected, projections of the assembly are generated in the WCS (the Global coordinate system).

14.6. Geometrical dimensions

Use this part of the dialog to define or modify a dimensioning style depending on the selected category:

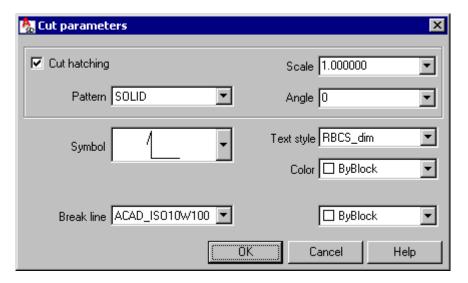
- single part
- assembly
- group

See also:

Geometrical dimensions (3D view)

14.7.Cut parameters

The Cut parameters dialog displays when you click Cut parameters in the Definition / modification of dimensioning style dialog.



If Cut hatching is selected, surfaces seen as cuts through element walls are hatched. Define hatching parameters:

- Pattern select an available pattern from AutoCAD®
- Angle select the value of the inclination angle of the hatching pattern
- Scale select a value that will elongate or shorten the hatching pattern

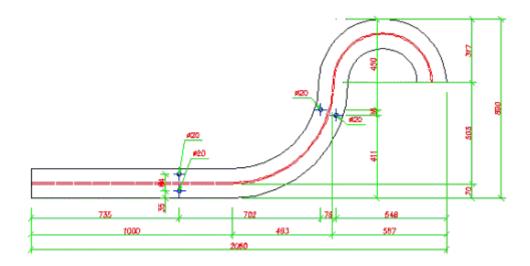
In the lower part of the dialog, it is possible to determine parameters of a cut through elements:

- Cut symbol
- Style and color of text
- Style and color of a break line

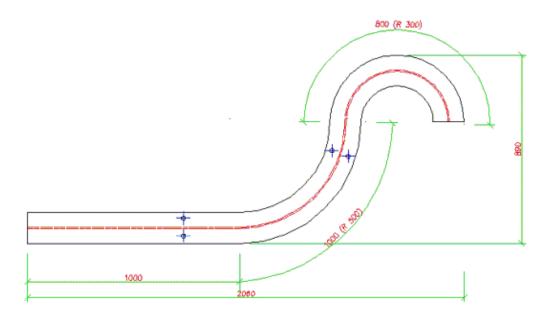
14.8. Dimensioning of curved parts

Specify how curved elements are displayed on the screen and dimensioned. Single parts:

 When Orthogonal is selected, curved parts display as standard objects so that their actual shape is shown and dimensions are generated in the orthogonal system (compare example dimensions generated with the use of this option in the drawing below).



• When Aligned to curve is selected, curved parts display as curved objects so that their actual shape is shown and dimensions are generated in the system adjusted to the profile curvature (compare example dimensions generated with the use of this option in the drawing below). Total dimensions are displayed in the orthogonal system (coordinate system of the curved part), and the remaining dimensions are generated in the longitudinal / transversal system. Longitudinal dimensions are always parallel to the profile axis (dimension values are calculated along the length of arc axis, and a radius value is given in parentheses); transverse dimensions are segments perpendicular to the profile axis and passing through the dimension point.



 When Developed is selected, object dimensions are displayed in the developed form in the orthogonal system. Click Options to open the Developed parts dialog.

Assemblies

- When Orthogonal is selected, assemblies display as standard elements so that their actual shape is shown and dimensions are generated in the orthogonal system.
- When Aligned to curve is selected, assemblies are displayed as curved elements so
 that their actual shape is shown and dimensions are generated in the system adjusted
 to the profile curvature. Total dimensions are displayed in the orthogonal system
 (coordinate system of the curved part), and the remaining dimensions are generated
 in the longitudinal / transversal system. Longitudinal dimensions are always parallel to
 the main part axis (dimension values are calculated along the length of arc axis, and a

radius value is given in parentheses). Total transverse dimensions are determined by cutting planes (external transverse dimensions are parallel to total transverse dimensions); internal transverse dimensions are segments perpendicular to the profile axis and passing through the dimension point.

Single parts

For the Orthogonal and Aligned to curve options, projections of the actual element shape are obtained, and elements are projected onto planes of the coordinate system of a curved part. The coordinate system of a curved part is a coordinate system used only in drawings, defined as follows:

- For profiles
 - If a profile includes linear segments, the longest segment is selected and the coordinate system of a curved part coincides with the coordinate system of this segment (as if it were a single simple profile).
 - If a profile does not contain linear segments, the coordinate system of a curved part may be defined as:
 - Method A the X axis of the coordinate system of a curved part is positioned along the chord of the longest (among all parts) arc segment
 - Method B the X axis of the coordinate system of a curved part is tangent to the chord of the longest (among all parts) arc segment at the beginning point

The method of defining the coordinate system of a curved part is selected in the Dimensioning Style Settings dialog:

- If Align arc profiles to the chord is not selected, the coordinate system is defined using method A.
- If Align arc profiles to the chord is selected, the coordinate system is defined using method B.

In both cases, the Y axis of the coordinate system of a curved part is positioned in the plane of part bending, irrespective of profile rotation.

For plates

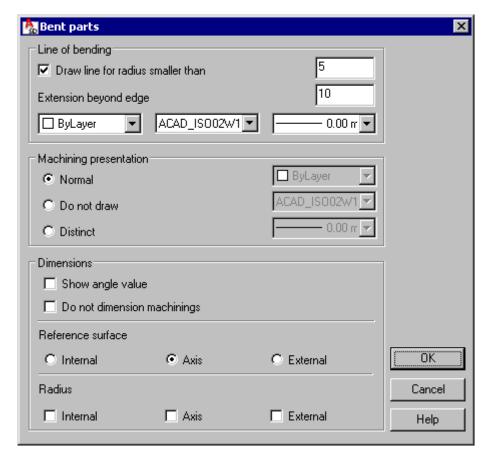
The coordinate system of a curved part is identical with the part UCS.

Assemblies

Assemblies are considered curved if the main part of an assembly is curved. For curved assemblies, projections of the actual element shape are obtained, and elements are projected onto planes of the coordinate system of a curved part (main part of an assembly). The coordinate system of a curved part is defined as for a single part.

14.9. Developed parts - options

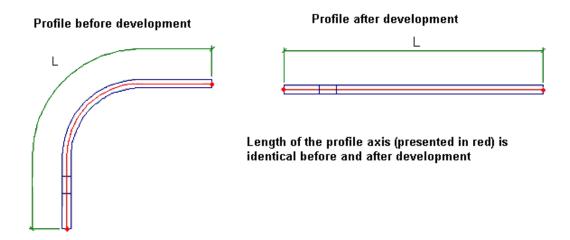
The Bent parts dialog displays after you click Options in the Dimensioning style settings dialog (the option is available for curved parts presented in a developed form).



The dialog includes the following fields:

- Line of bending use these options to define parameters of bending edges
- Machining presentation use these options to determine how to display machining in a drawing; selection lists in the right part of the field are accessible only when Distinct is selected.

The idea of element development consists in transforming the curve representing the element axis into a straight line in such a way so that the axis length and distances between points on the axis remain the same (see the drawing below).



The line of bending is positioned in the middle of each arc-shaped segment. Color, type, and thickness of the line are defined in the bottom part of the Line of bending field. The line of bending is displayed in a drawing if you select Draw line for radius smaller than. A radius value may also be specified (the line is usually drawn for small radiuses).

For Extension beyond edge, define a value of distance (extension of the line beyond the element edges on both sides of the element).

Under Machining presentation, select a presentation type (NOTE: the options below are not available in version 5.0):

- Normal if this is selected, all openings, cuttings, and so on, will be displayed in a drawing consistently with the style adopted for standard parts
- Do not draw if this is selected, only the profile shape will be displayed in a drawing (machinings will not be shown)
- Distinct if this is selected, machinings will be displayed in a drawing; a shape of developed machinings will be defined identically as for normal presentation type. In addition, use the selection lists to specify thickness, color, and line type parameters.

Curved parts are dimensioned in the same manner as rectilinear parts. You can switch off dimensioning of machinings that may be shown in a drawing of a developed part. If you select Do not dimension machinings, all dimension points defining machinings are disregarded and only total dimensions and dimensions to the line of bending are displayed.

The options in the lower part of the dialog refer to drawings of developed bent plates. These drawings display the original shape and dimensions of plates (without plastic deformations). Use these options to insert the following items in a drawing: bending lines, angle of bending, radius (external or to axis), dimensions, and machinings of a plate.

See also: Bending of plates

14.10. Geometrical dimensions (single part)

Dimension lines are positioned outside an element in the following order starting from the edge of a dimensioned element:

- 1. Elements located inside the dimensioned part
- 2. Drilled openings
- 3. Chamfers and cuts
- 4. Overall dimensions

This part of the dialog consists of 3 identical tabs on which the following parameters may be defined:

- Dimension format for all dimensions generated automatically in a view, the following dimension settings are available:
 - Relative chain the program generates a complete dimension chain with standard (relative) dimensions
 - Absolute chain the program generates a dimension chain which begins in the reference point; dimensions are positioned on extension lines - a dimension value indicates the coordinate of a dimensioned point with respect to the reference point
 - Relative + absolute chain the program generates an accumulative dimension chain and on the dimension line provides real (relative) dimensions of segments between the dimensioned points
 - To base each dimension is positioned on a separate dimension line attached to the reference point

Click Ref. point to open the Reference point dialog, where you can define an additional dimension point during generation of some dimensions (for example, absolute dimensions).

• Overall - these are the linear dimensions that display an overall size of a dimensioned part. These dimensions are generated if the option is selected, and they are always positioned along the horizontal or vertical axis (parallel to the plane of the element local coordinate system). Horizontal dimensions may be placed above or below the dimensioned element, and vertical dimensions are positioned to the left or to the right of the dimensioned element. It is possible to exclude horizontal and vertical dimensions from presentation. If Optimal is selected, the dimension line is positioned on this side where the longest edge of the dimensioned element is placed (so that extension lines on the edges do not need to be lengthened).

- Bevels and cuts these dimensions concern all the discontinuances on element edges (corners, cuts, chamfers, and so on), but do not pertain to drilled openings. There are 3 dimension options:
 - Chain
 - all dimensions are displayed on the common dimension line
 - lines are always positioned along the vertical and horizontal axes
 - dimension lines are presented on this element side where a discontinuance is found - dimension lines are drawn to reach the most distant corners (edges) of an element - identically as for overall dimensions
 - From edge
 - each dimension is presented on a separate dimension line
 - successive dimensions show distances from the common point to successive characteristic points of an element
 - dimension lines are presented on this element side where a discontinuance is found
 - o Aligned
 - each segment of a dimensioned element's edge is ascribed its own dimension line which is parallel to the dimensioned segment
 - in the case of arc segments, the chord is subjected to the dimensioning
- Internal recess these are the linear dimensions of recesses included inside the element contour:
 - Inside part
 - dimension lines are drawn inside the contour of a dimensioned element
 - all dimensions are presented on one dimension line (dimension chain) vertical or horizontal
 - the dimension line is extended to the closest edge of a dimensioned element
 - Outside part dimension lines are drawn outside the dimensioned element (the same rules are applicable as in the case of bevels and cuts)
- Openings in view these are the linear dimensions that indicate positions of drilled openings. Dimension points may be positioned at opening centers or at external points of the contour of a circle representing an opening; the Chain and From edge options are also available (as for bevels and cuts).
- Perpendicular openings these are the linear dimensions that indicate positions of drilled openings located in the plane perpendicular to the projection plane. The same rules apply here as for openings in view; the Chain and From edge options are available (as for bevels and cuts).
- Angular these are the dimensions displayed for the edges of a dimensioned element that are not parallel to the axes of the element local coordinate system
 - line with respect to which the dimension is to be determined (with respect to horizontal or vertical line)
 - o value of the angular dimension expressed as: angle value or edge inclination

Define arc dimensioning (these dimensions do not concern drilled openings and circles):

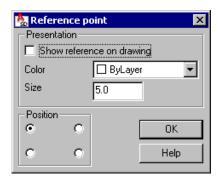
- Diameters for all circles (not arcs), a circle diameter is displayed. A dimension line coincides with the circle diameter positioned at an 45-degree angle to the horizontal axis
- Radial for all arcs and circles, the radius is displayed. Circles are dimensioned in this
 manner if the Diameters option is not selected. A dimension line coincides with the
 radius set at an 30-degree angle starting from the beginning point of the arc.

14.11. Reference point

The reference point is an auxiliary point positioned at one of the 4 points of the main part, and may be shown in a drawing as \bigcirc .

The reference point is used as an additional dimensioned point in generation of some dimensions (for example, absolute dimensions). It is positioned on one of the ends of the main part's edge parallel to its axis (by default, the Bottom, Left point).

Position, size, and color of the reference point can be configured in the Reference point dialog, which opens when you click Ref. point in the New dimensioning style dialog.



14.12. Geometrical dimensions (assembly)

Use the tabs in this part of the dialog to define the following parameters:

- Dimension format as for single parts
- Overall the option operates identically as in the case of single parts
- Part spacing these dimensions display mutual position of the assembly component parts. The dimensions are shown on external dimension lines (vertical or horizontal), and are generated parallel to the overall dimension lines. The following options are available:
 - Contour
 - Dimensions along the main axis (horizontal dimensions) beginning and end
 of a subpart is projected onto the direction parallel to the axis of the element's
 main part
 - Dimensions transverse with respect to the axis (vertical dimensions) beginning and end of a subpart is projected onto the direction parallel to the specified direction of dimensioning, and the reference point should also be projected on this line. If any of these dimension points is positioned on one of the edges of the main part, parallel to the axis of the main part, this point is not dimensioned; a given part is projected on the dimension chain positioned on this side where the edge closer to the dimensioned part is located.
 - o Axes
 - Dimensions along the main axis (horizontal dimensions) the point of intersection of the element axis and the main part's edge, parallel to the axis and the closest to the geometrical center of the dimensioned subpart.
 - Dimensions transverse with respect to the axis (vertical dimensions) significant only for elements whose axis is parallel to the axis of the main element (the axis is projected onto the direction perpendicular to it). A dimension is placed on the vertical dimension line; if the axis of a dimensioned subpart coincides with the axis of the main part, then the dimension is not generated.
 - o Point
 - Dimensions along the main axis (horizontal dimensions) one of the edge points of a subpart is projected onto the direction parallel to the main part's axis (a point positioned closer to the main part's edge is selected). If both edge points of a subpart are located on the main part's edge, the point closer to the reference point is dimensioned.
 - Dimensions transverse with respect to the axis (vertical dimensions) the first (counting from the reference point) point met on the subpart's edge is projected onto the direction parallel to the specified direction of dimensioning. The reference point should be projected onto this line, as well; if any of these dimension points is positioned on one of the edges of the main part, parallel to the main part's axis, this point is not dimensioned. A given part is projected on the dimension chain positioned on this side where the edge closer to the dimensioned part is located.

Optimal

 Dimensions along the main axis (horizontal dimensions) – selection depends on a subpart type

Plates perpendicular to the main part's axis, set orthogonally – the axes option Plates perpendicular to the main part's axis, set slanting - the point option Plates parallel to the main part's axis – the contour option Profiles perpendicular to the main part's axis – the point option Profiles parallel to the main part's axis - the contour option

 Dimensions transverse with respect to the axis (vertical dimensions) selection depends on a subpart type:

Plates perpendicular to the main part's axis, set orthogonally - the point option Plates perpendicular to the main part's axis, set slanting - the point option Plates parallel to the main part's axis - the contour option Profiles perpendicular to the main part's axis - the contour option Profiles parallel to the main part's axis - the contour option

Additionally, the following options are accessible (they parametrize the position of a dimension line):

With overall – dimension lines are always placed on the same side as overall dimension lines. If the position of overall dimensions is not defined, lines are positioned according to the Optimal option.

Optimal - dimension lines are placed on the element side that is closer to a greater number of dimension points.

- Part dimensions used to display only overall dimensions of subparts; dimensions are
 generated following the same rules as in case of the whole assembly. These lines are
 always generated as internal dimension lines, the orientation of which is determined in
 the local coordinate system of a dimensioned subpart or in the coordinate system of
 the main part of an assembly depending on the settings: part CS (Coordinate System)
 or assembly CS. If any of the dimensions generated by the algorithm is already shown
 on the external dimension line as an overall dimension, then it is not repeated.
- Openings in view used for dimensioning of opening positions with respect to the element's main part. Dimensions in the direction parallel to the main axis are placed on the external dimension line, whereas transverse dimensions are positioned on external or internal dimension lines depending on an opening position in a part. The Axes and Contour options are taken into account for dimensions transverse with respect to the main element's axis.

Dimensions along the axis: all the dimensions are projected onto common external dimension line that is parallel to the main element's axis.

Dimensions transverse with respect to the axis: these dimensions are placed on external or internal dimension lines depending on opening positions.

Dimensions on external dimension lines are generated for openings positioned on transverse edges of an assembly (in endplates). External dimension lines are positioned parallel to the direction specified for overall dimensions, and are always attached to the element's overall dimension.

Internal dimension lines are generated for openings positioned along the main element's axis. They are always placed perpendicularly to the main element's axis and as close as possible to dimensioned openings (in such a way so that they do not overlap with other elements in a drawing). Internal dimension lines are attached to the main element's axis or to one of external edges of the main part that is parallel to the axis (an edge positioned closer to a greater number of dimensioned openings is selected).

- Perpendicular openings –works the same as the Openings in view option, but if both options are selected, all points are projected onto a common dimension line.
- Angular the following parameters can be specified:
 - External the dimensioning concerns external edges of the main part and of subparts placed outside the main part's contour, positioned unorthogonally with respect to the coordinate system of the main part. An acute angle is always shown, the base of which is a vertical line.

- Internal the dimensioning concerns edges of subparts placed within the main part's contour, positioned unorthogonally with respect to the coordinate system of the main part. An acute angle is always shown, the base of which is an element's edge parallel to the axis of the main part.
- Both dimensions are generated on the basis of both the internal and external options.

The options of arc dimensioning operate identically as in the case of single parts.

Define the Depth of a structure for which assemblies will be presented in drawings:

- Relative with respect to a value of the 'depth' of a whole structure (100% total depth)
- In absolute units in current units

14.13. Geometrical dimensions (group)

Use the tabs in this part of the dialog to define the following parameters:

- Overall the option operates identically as in the case of single parts.
- Axial dimensions dimensions are generated as a dimension chain showing axial dimensions of a presented group. Dimension points are generated at points of axis intersection (for groups, axes of main assembly parts are displayed; therefore, axial dimensions are determined only for the visible axes). Available options:
 - Orthogonal dimension chains are positioned along vertical or horizontal edges of the view.
 - Aligned dimensions are positioned as parallel to the dimensioned segment, and are not extended to the most distant points of a group. Additional dimension points are generated at the beginning and end of the group where there are no axis intersections.
 - Both both methods described above are applied.
- Assembly dimensions dimensions display the entire dimensions of assemblies included in a group, and are generated identically as overall dimensions for each assembly separately. Available options:
 - o Chain all dimensions are projected on a common external dimension line (chain) positioned vertically or horizontally.
 - From edge each dimension is displayed on a separate dimension line, each of which is parallel to the axes of the local coordinate system. They are located on the side of a dimensioned element that is positioned closer to dimension points.
 - Aligned dimensions are set in a parallel position with respect to the main assembly part (in the local coordinate system of the assembly), and are displayed on the external side of the assembly (in relation to the group center).
- Angular the option operates identically as in the case of single parts.

The options of arc dimensioning operate identically as in the case of single parts.

14.14. Geometrical dimensions (3D view)

All dimensions generated automatically are positioned in one of the (orthogonal) planes belonging the local coordinate system of a part or group.

For an object being dimensioned (part, assembly, or group) the software defines a rectangular prism circumscribed on the selected part (the prism is perpendicular to the local coordinate system of the object). Dimension lines are positioned in one of the planes of the prism, so that they do not conceal the object. In the upper left of the dialog are options that refer to these planes in which dimension lines are positioned: Distance to the first dimension line and Distance between dimension lines.

Single part or Assembly

For a single part or an assembly, the Dimensions field includes options that let you select dimensions and direction to be generated automatically.

Group

Available options:

Presentation - choose a method of displaying a part in the drawing:

- o Real a full drawing of a part with all details is displayed
- o Simplified profile fillets are ignored
- Scheme profiles are displayed as lines

Specify settings for color, type, and thickness of a line displaying a profile.

Dimensions

For groups, the Dimensions field comprises options enabling selection of dimensions and direction to be generated automatically (for total dimensions, element axes, and workframe axes).

14.15. Additional texts

If the Comment option is selected, a title (name) is added to the views of a dimensioned object. The title may be text (comprising one or several lines) containing variables or any user-defined character string. At the bottom of the Comments field are options that, when selected, add a variable to the defined comment at the cursor position. The complete text of the comment is displayed in the edit field.

Available comment variables:

%%Pos - if this is selected, a position description is added.

%%Name - if this is selected, a profile or plate name (for example, IPE 100) is added.

%%Scale - if this is selected, a scale (for example, 1:20) is added.

%%Vname - if this is selected, an automatic name of a view or section is added.

%%Length - if this is selected, profile or plate length expressed in the current project units is added.

%%Mat - if this is selected, name of a part material is added.

%%Quant - if this is selected, a number of elements of a presented position is added.

%% Weight - if this is selected, a calculated part weight is added.

%%Family - if this is selected, the name of a family to which the part belongs is added.

%%Surf - if this is selected, a type of surface finishing of a structure element is added.

You can specify a comment in a drawing that includes:

- Text style
- · Position of a text in a view
- Text aligning (align right, align left, center)
- Color of the comment text

14.16. Styles of descriptions and symbols

In this part of the dialog, select styles for the following descriptions and symbols (designations) that display in final drawings:

- Openings/bolts
- Parts
- Assemblies
- Elevation marks
- Weld symbols
- Workframe axes



The number of symbols depends on the selected category of dimensioning style (single part, assembly, or group).

Clicking one of the (...) buttons to the right of the symbols opens the Styles of descriptions dialog with the symbol category selected.

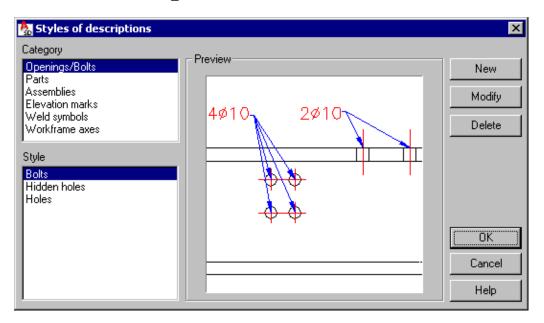
15. DESCRIPTION STYLES

15.1.Description styles

Use the Styles of descriptions dialog to define description styles for elements. The dialog displays after you click a (...) button in the Dimensioning style settings dialog.

To being defining a description, open the Styles of description dialog from:

- Menu: Steel / Styles / Description styles
- Ribbon: ASD Drawings / Styles / Description styles
- Command line: RBCS DESCRIPTION.



This dialog operates in a similar manner as the Dimensioning styles dialog. A set of description styles is saved to a DWG format file or in a template (a DWT file).

To display styles defined for a category type, select an option under Category. When you highlight an option for Style, all styles defined for the selected category are displayed. A preview of a selected style is shown in the Preview field.

Available categories:

- Openings/bolts
- Parts / Assemblies
- Elevation marks
- Weld symbols
- Workframe axes

One style defined for each category is the default style, which will be applied during manual (not automatic) definition of symbols in a drawing.

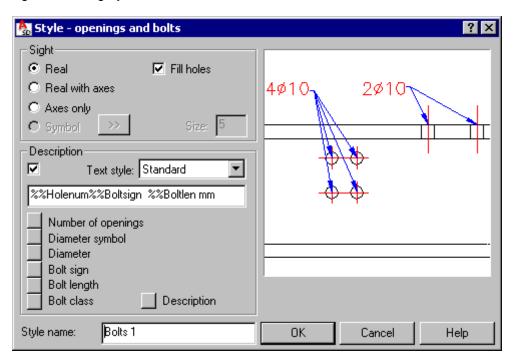
Use the buttons in the dialog:

- New click this to open a dialog where you can define a description style for a selected category. The name of thye new style and all its settings are inherited from the style currently selected.
- Modify click this to open a dialog where you can modify a description style of a selected category.
- Delete click this to delete the style currently selected from the list of available styles for a given category.

15.2. Styles of descriptions - Openings/Bolts

Use the Openings/Bolts dialog to define a new style or modify an existing style of opening or bolt description.

The dialog displays after you click New or Modify in the Styles of descriptions dialog when the Openings/Bolts category is selected.



Define parameters:

- Style name use this edit field to specify a name for the style
- Under Sight, select a method of opening generation in a drawing
 - Real real opening edges are displayed (openings in view and perpendicular openings)
 - Real with axes real opening edges are displayed (openings in view and perpendicular openings) with opening axes
 - Symbol when this is selected, openings in view and perpendicular openings are displayed as symbols. Click the >> button to extend the dialog to include a library of symbols available
 - Fill holes if this is selected, an opening for the Real or Real with axes option will be displayed as shown in the drawing below



Use the buttons in the expanded dialog:

- Edit opens a dialog in which you can modify a selected symbol.
- Add opens a dialog where you can define a symbol (as an AutoCAD block).
 An insertion point for a defined block is treated as a center of a drilled opening (top view) or as an end of the opening axis (side view).
- Delete deletes a selected symbol

Opening symbols must be defined as a pair: opening in view and perpendicular opening. For Size, specify the size of a symbol in a final drawing, expressed in the currently applied project units (for a symbol displayed in a top view). A symbol in a side view is scaled proportionally to obtain a specified size, but symbol length is adjusted to the thickness of a drilled part.

- The Preview shows an opening.
- Under Description, define a manner of opening description. Descriptions of openings are generated if Description is selected.

For Text style, select from the list of text styles defined in a DWG format file; the edit field below the list displays a description (text) syntax. Text may comprise any user-defined character string. At the bottom of the Description field are options that, when selected, add a variable to the defined comment at the cursor position:

%%Holenum - if this option is selected, a number of openings in the opening group is added.

%%Diasym - if this is selected, a \varnothing symbol is added.

%%Dia - if this is selected, a value of opening diameter is added.

%%Boltsign - if this is selected, a bolt symbol is added.

%%Bltlen - if this is selected, bolt length is added.

%% Boltclass - if this is selected, a bolt grade is added.

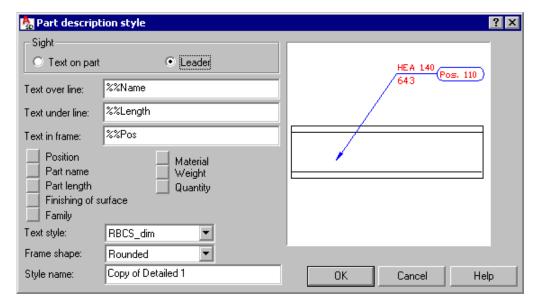
%%Desc - if this is selected, a bolt description is added.

• If Groups on common leader is selected, one description (one leader) is generated for all openings in a group during dimensioning. If this is not selected, descriptions (leaders) are generated for each opening.

15.3. Styles of descriptions - Parts/Assemblies

Use the Part description style dialog to define a new style or modify an existing style of part or assembly description.

The dialog displays after you click New or Modify in the Styles of descriptions dialog when the Parts or Assemblies category is selected.



Define parameters:

- Style name use this edit field to specify a name for the style
- Under Sight, select a method of opening generation in a drawing
 - Text on part when this is selected, you can define a description in the Text field.
 Use the Frame shape field to specify a frame for the text. Text is placed within the
 contour of a dimensioned element (positioned to avoid overlapping of elements
 displayed in a drawing).
 - Leader when this is selected, there are 3 edit fields available: Text over line, Text under line, Text in frame. Define each of them using any user-defined character string.

There are options available under these fields that, when selected, add a variable to the defined comment at the cursor position:

%%Pos - if this is selected, description of a position is added.

%%Name - if this is selected, a profile or plate name is added.

%%Length - if this is selected, profile or plate length is added.

%%Mat - if this is selected, a name of a part material is added.

%% Weight - if this is selected, a calculated weight of a part is added.

%%Quant - if this is selected, a number of parts is added.

%%Surf - if this is selected, a type of surface finishing of a structure element is added (available in the Part description style dialog; it is not available in the dialog for the assembly description style).

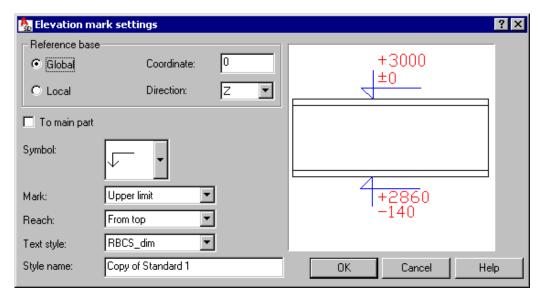
%%Family - if this is selected, the name of a family to which the part belongs to is added.

- Text style select a font to be applied in descriptions
- Frame shape select a frame shape for a leader: rectangular, rounded, or none (without frame)

Text in leaders is positioned outside dimension lines in order to avoid overlapping of leaders with other drawing elements or leaders with other leaders.

15.4. Styles of descriptions - Elevation marks

Use the Elevation mark settings dialog to define a new style or modify an existing style of description of elevation marks; the elevation mark consists of a symbol and a coordinate. The dialog displays after you click New or Modify in the Styles of descriptions dialog when the Elevation marks category is selected.



Define parameters:

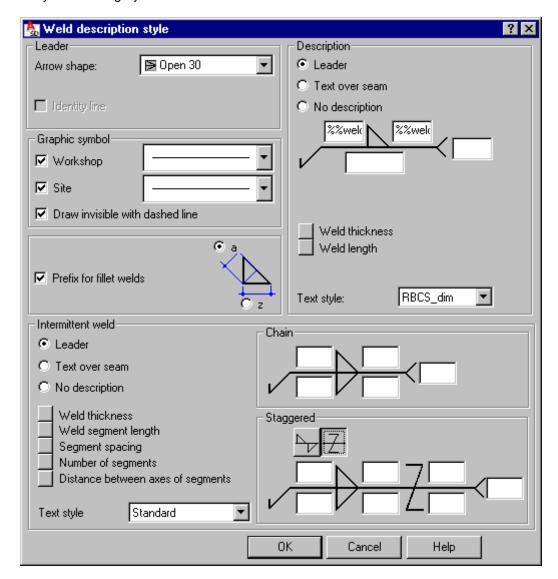
- Style name use this edit field to specify a name for the style
- Under Reference base, choose the coordinate system (global coordinate system of a model or local coordinate system of an element) that is the basis for defining elevation reference (coordinate) and direction.
- If To main part is selected for assemblies and groups, limits are determined for a main part of the group and not for a group as a whole.
- For Symbol, select an elevation symbol provided with the software.
- For Mark, select a part of the element edge (upper limit, lower limit, or both) on which an elevation mark is to be generated. Symbols are generated automatically at the most distant point along the selected direction.
- For Reach, define a point of connection between a symbol and an edge of a dimensioned element (from top, from bottom, or aligned).
- Text style select a font to be applied in descriptions.

Symbols are located near edges of a dimensioned element (inside dimension lines).

15.5. Styles of descriptions - Weld symbols

Use the Weld description style dialog to define a new style or modify an existing style of weld symbol description.

The dialog displays after you click New or Modify in the Styles of descriptions dialog when the Weld symbols category is selected.



Define parameters:

- Style name use this edit field to specify a name for the style
- Arrow shape select a shape from the list; a user-defined shape is available, as well
- Text style select a font to be applied in descriptions

The weld symbol is composed of the following elements:

- Leader ended with an arrow
- Line
- Weld symbol
- Weld thickness and length (expressed in mm) click the corresponding buttons to automatically place the dimension in the active text field.
- Additional designations

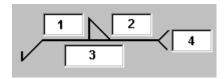
The following rules apply to generation of weld descriptions:

- All visible welds are given a description.
- An arrowhead that indicates an edge is attached to the center of gravity.

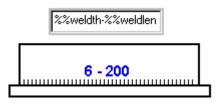
- The arrow line may intersect an element edge or a dimension line.
- Weld description and extension line may not overlap with the element edge or a dimension line.

Under Description, configure a weld description:

- Leader the description may be positioned in the following locations (numbering corresponds to the drawing below):
 - 1. Above the extension line before the weld symbol
 - 2. Above the extension line behind the weld symbol
 - 3. Under the extension line
 - 4. Behind the extension line



 Text over seam - a description specified in the edit field will be placed above the weld seam (any character string with the possibility to use variables, such as %%weldth-%weldlen) - see the drawing below.



No description (only a weld is drawn)

In all locations, variables may be used (as in bolt descriptions). Two variables of weld description are available:

%%weldth - weld thickness

%%weldlen - weld length

Descriptions are generated for all welds displayed in a selected projection of an assembly or group. If there are several welds of the same parameters located close to each other, it is possible to generate one description on several leaders.

If Draw invisible with dashed line is selected, invisible weld edges (concealed by other elements) are displayed in a drawing by means of a dashed line. If this option is not selected, invisible edges are not drawn.

If a circumferential weld forms a closed contour in a model, the software automatically identifies such welds in all drawings. An additional symbol (a circle) is added to the weld symbol in the same place as the site weld symbol.

You can use a line to identify a weld face in 2D drawings (when the Identity line option is selected). The identity line may be placed under or above an extension line, and is denoted with a dashed line of the same thickness as the extension line.

If Prefix for fillet welds is selected, it enables:

- Selection of the **a** option for a weld description, the software reads the thickness specified in the weld definition in a model, and enters a prefix before the weld thickness (a5).
- Selection of the **z** option for a weld description, the software calculates the value $z = a\sqrt{2}$ and enters e.g. z7 as the weld thickness.

If Prefix for fillet welds is not selected, only the weld thickness (from a model definition) is provided.

For 2D drawings, it is possible to display a graphical symbol of a weld using one of the lines shown below (the options are available for selection under Graphic symbol):

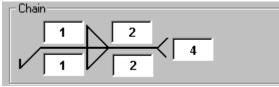


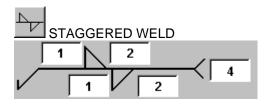
These lines can be configured separately for workshop welds and site welds. Line thickness and color are assumed to be the same as for the extension line.

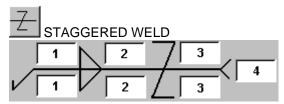
In the lower part of the dialog, you can specify description parameters for intermittent welds:

- Leader
 - A description may be positioned in the following locations (numbering of the description locations corresponds to the numbering in the drawing below):
 - 1. %%weldth
 - Metric units: %%NumOfSeg x %%LengthOfSeg Imperial units (USA) %%LengthOfSeg - %%DistBeetwAxis
 - 3. (%%LengthOfSpace)
 - 4. Any character string

CHAIN WELD







- Text over the weld seam
 A description (any character string) specified in the edit field will be positioned over the weld seam.
- No text (only a weld is displayed)

In all locations, variables may be used (as in bolt descriptions). Several variables of weld description are available:

%%weldth - weld thickness

%%lengthofseg - length of a weld segment

%%lengthofspace - spacing between segments of a weld

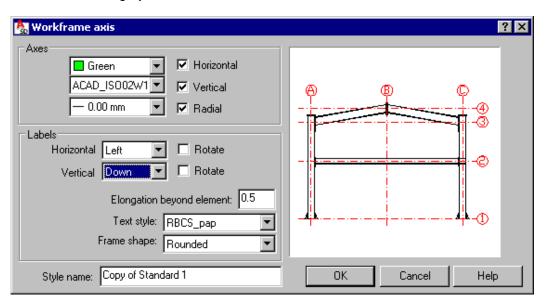
%%distbetwaxis - distance between axes of weld segments

%%numofseg – number of segments

15.6. Styles of descriptions - Workframe axes

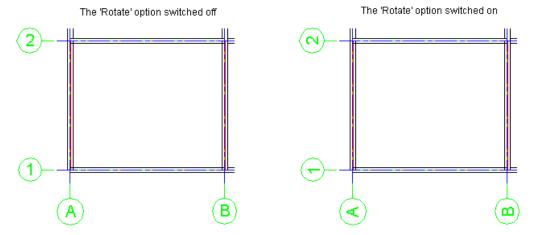
Use the Workframe axis dialog to define a new style or modify an existing style of describing workframe axes.

The dialog displays after you click New or Modify in the Styles of descriptions dialog when the Workframe axes category is selected.



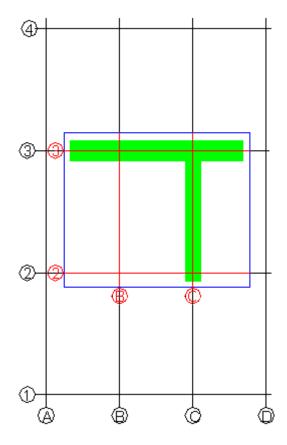
Define parameters:

- Under Axes, specify color, style, and line thickness parameters for workframe axes. If
 Horizontal, Vertical, and Radial are selected, the corresponding workframe axes will
 be displayed in drawings. If any of the options are not selected, the corresponding axis
 type will not display in drawings.
- Under Labels, specify description parameters for workframe axes:
 - Vertical axes may be displayed with labels (above the axis, under the axis, or on both sides) or without labels. If Rotate is selected, axis labels are rotated by 90 degrees (see the drawing below).
 - Horizontal axes may be displayed with labels (on the left side of the axis, on the right side of the axis, or on both sides) or without labels. If Rotate is selected, axis labels are rotated by 90 degrees (see the drawing below).



 Elongation beyond element - workframe axes are contained within a rectangular contour, in which a dimensioned group is inscribed. The size of this contour is determined by a rectangle circumscribed on the group contour increased by the value defined in this field (this value is added to each side). Rectangle sides are parallel to view edges, as shown below:

- o Green represents an element scheme
- Blue represents element borders with a margin added on each side (a value of elongation beyond element defined in the above dialog)
- Black represents the entire workframe
- Red indicates which part of the workframe will be shown and where labels will be placed



- Text style select a font to use for workframe description; the list displays all AutoCAD® text styles available in the current project
- Frame shape –select the label shape (rounded, rectangular, or none) for a workframe axis description
- Style name use this edit field to specify a name for the style

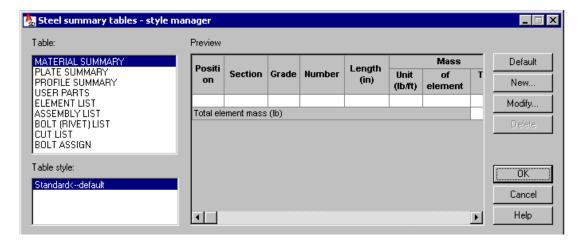
16. STEEL SUMMARY TABLES

16.1.Steel summary tables - style manager

Use this option to define/modify tables used to prepare steel tables.

To begin working with tables, open the Steel summary tables - style manager dialog from:

- Menu: Steel / Tables / Styles
- Ribbon: ASD Drawings / Settings / Table styles
- On the command line, enter RBCS_LISTTEMPL.



Available steel summary tables, listed under Table:

- Material summary
- Plate summary
- Profile summary
- List of profiles by section type
- User parts
- Element list
- Assembly list
- · Bolt (rivet) list
- Cut list
- Bolt list
- Mark list.

For each table type, the standard table style (displayed under Table style) has been defined in the software. After the table type and table style are selected, the current table view is displayed in the Preview field.

Use the buttons on the right of the dialog:

- Default click this to restore the default (standard) table layout (description style).
- New click this to open the Definition of new steel summary table style dialog, where
 you can define a new style of a selected table type (based on the style that already
 exists).
- Modify click this to open the Modification of steel summary table style dialog, where
 you can make changes in a selected table type and table style.
- Delete click this to delete a highlighted table style from the list of styles available in the Table style field.

16.2.Definition/modification of steel summary table style

Use the Definition of new steel summary table style dialog to define a new style or to modify an existing style of a table type.

The dialog displays after you click New in the Steel summary tables dialog (the Modification dialog displays after you click Modify).

The dialog consists of 4 tabs:

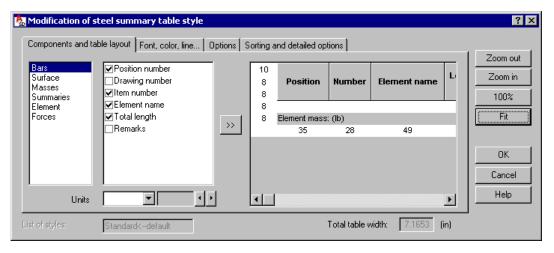
- · Components and table layout
- Font, color, line
- Options
- Sorting and detailed options.



If one of the steel summary table types (Assembly list) is selected, there is also a Syntax of summary line tab available.

16.3. Components and table layout

The image below displays the Components and table layout tab of the Definition/modification of steel summary table style dialog.





The options on this tab depend on a table type selected in the Steel summary tables - style manager dialog. The example above displays the options available after selecting the element list.

At the bottom of the dialog, for Style name, enter a name of the table style defined (when modifying the table style, the Style name field is inaccessible). To define/modify a table style:

- Select a set of table components (in the example shown, they include Bars, Surface, Masses, Summaries, Element, and Forces).
- In the next field, select the components to be included in the table
- Click the >> button.
 The right part of the dialog displays the defined table layout.

Specify additional parameters:

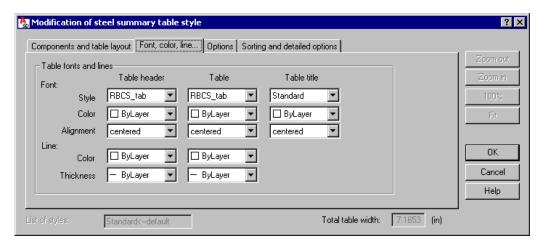
- Total table width read-only field that displays the width of the defined table; determined in the software.
- Zoom control buttons (Zoom in, Zoom out, 100%) that let you change the zoom factor of the table displayed.

Using the tables, you can:

- Add blank lines at the beginning and end of a table
- Add blank columns in tables
- Define table dimensions (column width and line height) dimensions of the column width and the line height are displayed under the table and on the left of the table, respectively.

16.4. Font, color, line

The image below displays the Font, color, line tab of the Definition/modification of steel summary table style dialog.

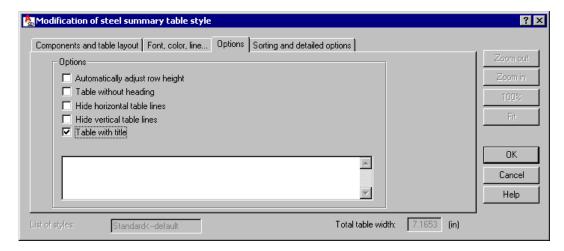


Use the available options to specify:

- Font style, color, and alignment used in the table header and in all table cells
- · Line thickness and color
- Font style, color, and alignment used in the table title (if Table with title is selected on the Options tab)

16.5.Options

The image below displays the Options tab of the Definition/modification of steel summary table style dialog.

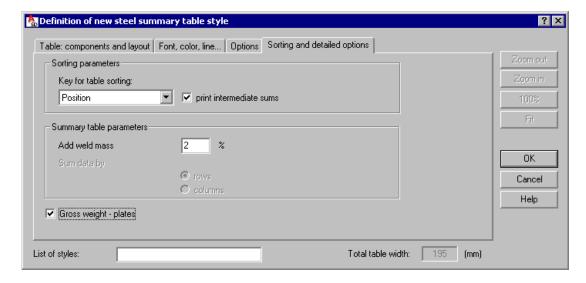


Specify options:

- Automatically adjust row height if this is selected, the height of table rows will be automatically adjusted to the size of symbols of element shapes displayed in the table.
- Table without heading if you select this, the table will be generated without a header.
- Hide horizontal table lines if this is selected, horizontal lines will not be displayed in the table.
- Hide vertical table lines if this is selected, vertical lines will not be displayed in the table.
- Table with title if you select this, the edit field in the lower part of the dialog becomes available and you can enter a table title. Use the Font, color, line tab to define a table title font.

16.6. Sorting and detailed options

The image below displays the Sorting and detailed options tab of the Definition/modification of steel summary table style dialog.



Specify parameters:

- Sorting parameters
 - Key for table sorting this is available when Steel Summary table is selected, and it lets you sorting the steel summary table by position, steel grade, or section (by default, it is sorted by position numbers).

- Print intermediate sums if this is selected, the table displays individual masses
 of individual elements of a steel structure (the total mass of all elements is
 displayed at the end of a table).
- Summary table parameters
 - Add weld mass define a value (expressed in percentage) that will be added to the total mass (for example, if you specify 2%, the total mass will be multiplied by the coefficient 1.02).

At the bottom of the dialog is the Gross weight - plates option. If this is selected, the weight of plates will be calculated as a weight of a plate sheet from which the plate is made (without considering machinings). If this is not selected, the weight of plates will be a net weight that includes all machinings of plates (chamfers, drilled openings, and so on).

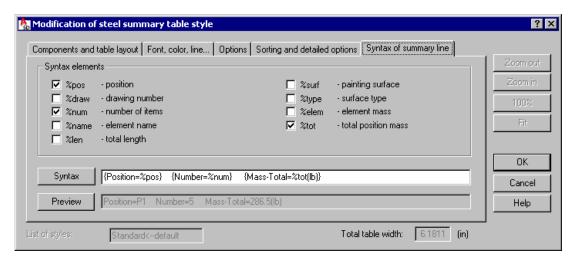
16.7. Syntax of summary line

The image below displays the Syntax of summary line tab of the Definition/modification of steel summary table style dialog.



NOTE:

The tab is available only for the Assembly list type of steel summary table.



Use the options on this tab to define a syntax of the first line (heading) of the assembly list table that contains the summary of assemblies. You can access the mechanism of the arbitrary composition of syntax and contents of the summary line.

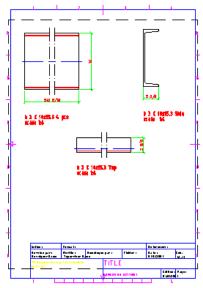
What you select under Syntax elements is displayed in the Syntax field. The Preview displays the description of the summary line resulting from the defined syntax. This description is based on fixed numeric values and responds to changes in the preferences (modification of unit, precision).

Available syntax element variables:

- position %pos %draw - drawing number %num - number of items %name - element name %len total length %surf painting surface %type surface type %elem - element mass %tot - total position mass

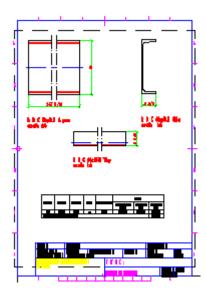
16.8. Example of generating and setting a table style

Creating tables of the bill of materials will be illustrated using automatic positioning and generating printouts. After positioning and generating printouts for position b2, the drawing in the figure below displays.



To generate a table of the bill of materials in the displayed drawing:

- 1. Click the Bill of materials option
- 2. In the drawing area, click to specify the insertion point of the table.



- 3. Select the table, right-click, and click Object properties.
- 4. In the Modification of selected table dialog:
 - On the Selection of table style tab, change the height of table rows.
 - On the Options tab, select Table with title.
 - In the edit field in the lower part of the dialog, enter a name (for example, Steel table position b2).
 - · Click OK to update the table.

17. PRINTOUT

17.1. Table printout manager

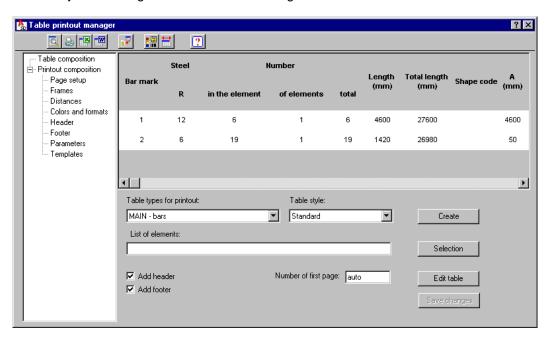
Use this option to define/modify the printout layout for tables used to prepare steel summary tables.

To begin defining a printout layout, open the Table printout manager dialog from:

- Menu: Steel / Tables / Table Printout Manager (steel part) or Reinforcement / Reinforcement table / Table Printout / Export / Edit (reinforcement part)
- Ribbon: ASD Drawings / Tables / Table Printout Manager (steel part) or ASD -Reinforcement / Reinforcement table / Table Printout / Export / Edit (reinforcement part)
- Toolbar: Table Printout Manager (steel part) or Table Printout / Export / Edit
 (reinforcement part)
- Command line: RBCS_LISTPRINT (steel part) or RBCR_LIST_EXP (reinforcement part).

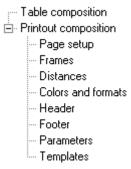


If you select Automatic table update on the Structural Detailing tab of the Options dialog, reinforcement tables in AutoCAD® Structural Detailing - Reinforcement will be updated automatically after changes are made in a drawing.



The Table printout manager dialog has 2 primary parts:

On the left is the selection tree (see the drawing below) from which you can select one
of the printout manager options



- To the right of the selection tree are the parameters that relate to the option you selected in the selection tree. The top part of the dialog shows the layout of a selected table type.
- Use the icons at the top of the dialog:

Printout preview - click this to open a preview of a table printout; return to the dialog by clicking Close

Print table - click this to start printing a table

Save table - click this to open the dialog in which a table may be saved in MS Excel®:

- *.CSV (Comma Separated Values) format files which are text files
- *.XLS format files this saving method fully reflects table settings that can be seen in a preview window

Save table (MS Word) - click this to open the Save As dialog, where you can save a table in an MS Word file with the specified name

Save graphical settings - click this to save current settings of the printout manager

Automatic adjust of column width to header text - click this to adjust the width of table columns to the length of table column names

Help - click this to open Help



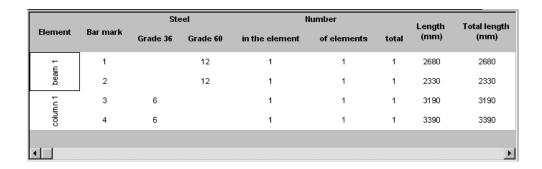
For AutoCAD® Structural Detailing – Reinforcement, an additional icon lets you insert a table into an edited drawing.

Although a table shape (cell height, column width) depends directly on a defined table style, you can determine the shape.

In addition, there is a context menu with the following options:

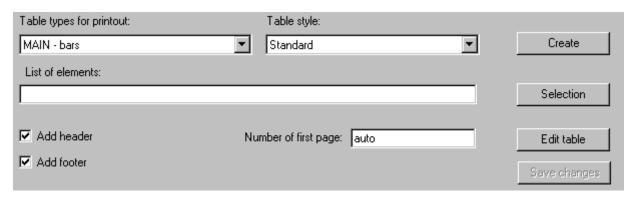
- Table cells group (merges several table cells into one cell) or ungroup
- Text orientation vertical, horizontal
- · Adding or deleting a column in the table
- · Adding or deleting a row in the table

The example below shows a table layout after adding a new column at the beginning of the table, merging several table cells, and changing text orientation to vertical. The table displays data for a few positions of structure element reinforcement.



17.2. Table composition

After you select Table composition from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



For steel structure elements, choose a table type:

- Material summary
- Plate summary
- · Profile summary
- User parts
- Element list
- Assembly list
- Bolt (rivet) list
- Cut list
- Bolt assign

For RC structure elements, choose a table type:

- Main (reinforcing bars)
- Summary (reinforcing bars)
- Detailed (reinforcing bars) this requires graphically selecting the distribution varying linearly / surface bar distribution or entering a number of reinforcement position
- Main (elements) see the description of the Element manager dialog
- Main (wire fabrics)
- Summary (wire fabrics)

For each table type, you can choose a table style defined previously for the selected table type.

Click Create to generate a table of a given type based on the current element list (if the list is empty, a printout of a whole structure is made) in the format of the active table style (see: Styles applied in tables).

The List of elements field displays numbers of the elements included in a table. Clicking Selection closes the dialogs and lets you graphically select elements to be included in the table.

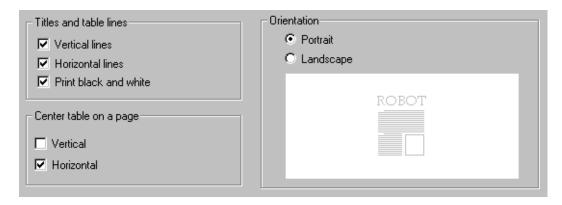
Clicking Edit table lets you graphically indicate the table to be edited. Changes made in a table may be saved after you click Save changes.

Use the options at the bottom of the dialog:

- Number of first page specify a number of the first page to be printed
- Add header if this is selected, a defined header will display on a printout
- Add footer if this is selected, a defined footer will display on a printout

17.3. Page setup

After you select Page setup from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.

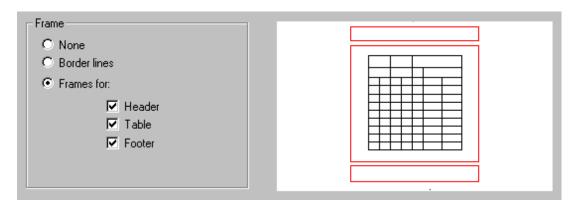


Determine the manner of table presentation:

- Under Titles and table lines:
 - Vertical lines if this is not selected, no vertical lines are displayed in a table
 - Horizontal lines if this is not selected, no horizontal lines are displayed in a table
 - Print black and white if this is not selected, a table is printed with the defined colors applied
- Under Center table on a page, specify how a table is to be centered (horizontally, vertically, or both)
- Under Orientation, define paper orientation (Portrait the longer side of a paper sheet is vertical or Landscape the longer side of a paper sheet is horizontal)

17.4.Frames

After you select Frames from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.

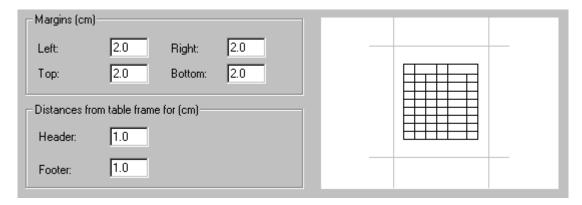


Determine the manner of table framing:

- None no separation between a header/footer and table
- Border lines includes lines that separate header and footer from the table
- Frames for select to include frames for the header, the table, the footer, or any combination of the three

17.5. Distances

After you select Distances from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



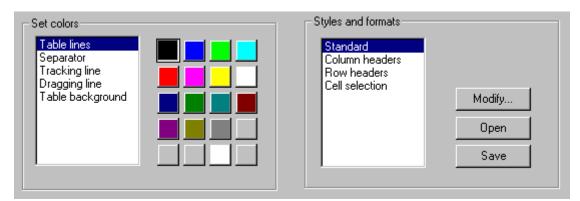
Under Margins, specify left, right, top, and bottom page margins.

Under Distances, define distances between the table frame and the header or footer.

The size of a header and footer is calculated automatically in the software; the parameters mentioned depend on a size of the applied font, size of a drawing with the company logo, and number of lines required in a header or footer.

17.6.Colors and formats

After you select Colors and formats from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



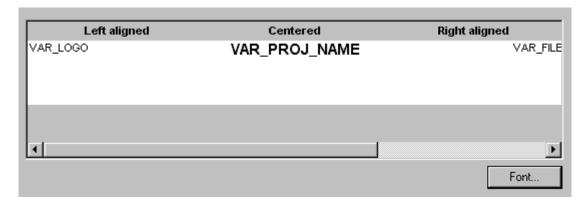
Under Set colors, specify the color of:

- Table lines
- Separator
- Tracking line
- Dragging line
- Table background

Under Styles and formats, you can select formats and styles applied in the following table elements: table column headers, table row headers, and text contained within a table. ClickModify to open the dialog whereyou can format font, font color, and alignment method for the selected table elements.

17.7.Header

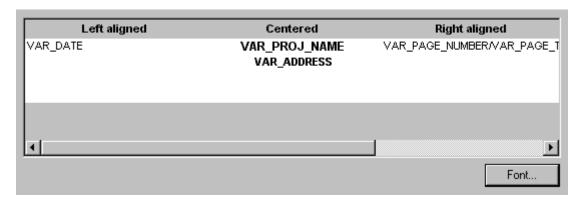
After you select Header from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



In order to change the layout of the printout header, select the table field you want to change, and then select a value from the list. To select a font for a particular field, position the cursor in the field, and click Font. In the dialog that opens, you can choose the font to be applied.

17.8.Footer

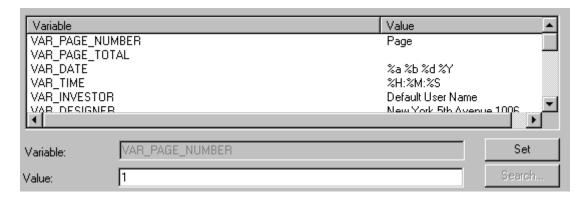
After you select Footer from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



In order to change the layout of the printout footer, select the table field you want to change, and then select a value from the list. To select a font for a particular field, position the cursor in the field, and click Font. In the dialog that opens, you can choose the font to be applied.

17.9.Parameters

After you select Parameters from the selection tree in the left part of the Table printout manager dialog, the options shown below display on the right.



The dialog contains all the variables defined in the system and their names.



In order to confirm a variable and its modifications, click Set.

Variables used for formatting the header and footer of the printout:

VAR_PAGE_NUMBER - lets you assign a current printout page. If VAR_PAGE_TOTAL is used, text assigned to the variable will precede the total number of pages (for example, Page 1, Page 2, and so on)

VAR_PAGE_TOTAL - value of this variable indicates the total number of printout pages. Text assigned to it may be preceded by a current page number, if the VAR_PAGE_NUMBER variable is used.

VAR_DATE, VAR_TIME - these variables may be assigned any text and combination of the key words displayed below (prints the current date/time on a printout):

%A - full week day name (Monday)

%a - abbreviated week day name (Mon)

%B - full month name (January)

%b - abbreviated month name (Jan)

%c - standard presentation of date and time

%d - month day (01-31)

%H - time (24-hour clock) (00-23)

%I - time (12-hour clock) (01-12)

%j - successive day of the year (001-366)

%M - minute (00-59)

%m - month (01-12)

 $\mbox{\ensuremath{\%p}}$ - local equivalent of the English abbreviations AM / PM

%S - second (00-59)

%U - successive week of the year (first day - Sunday) (00-53)

%W - successive week of the year (first day - Monday) (00-53)

%w - day of the week (0-6, Sunday is denoted by 0)

%X - standard time representation

%x - standard date representation

%Y - year and century

%y - year without the century specified (00-99)

%Z - time zone name

%% - percent mark

Standard date representation is the following string of variables: %a %b %d %Y Standard time representation is the following string of variables: %H:%M:%S Standard date and time representation is the following string of variables: %a %b %d %H:%M:%S %Y.

The remaining variables listed below do not contain other values than user-defined text. Their names serve only for the purpose of convenient classification while formatting.

The following printout variables may be used in AutoCAD® Structural Detailing - Steel:

VAR_POS - position name

VAR_SCALE - drawing scale

VAR_PART - part name

VAR_MATERIAL - part material

VAR_LENGTH - part length

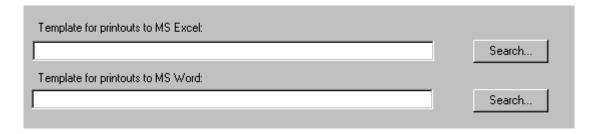
VAR_WEIGHT - part weight

VAR PAINTAREA - painting area

VAR QUANTITY - number of items of a given position in the project

17.10. Templates

After you select Templates from the selection tree located in the left part of the Table printout manager dialog, the options shown below display on the right.



You can select templates for printing tables to the following programs:

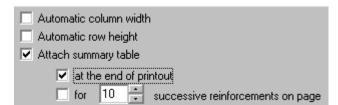
- MS Word®: DOT format files
- MS Excel®: XLT format files

In the edit fields, specify file names with a full access path. After you click Search, you can specify a template file from the computer hard disk.

In AutoCAD® Structural Detailing - Reinforcement, there are additional options available for printouts to MS Word®. They let you insert an additional summary table that includes information about the reinforcement (if Attach summary table is selected). You can set the table either at the end of the entire printout or for several successive reinforcements on a page.

Additional options:

- Automatic column width when this is selected, widths of table columns are automatically adjusted to fit the length of names of table columns.
- Automatic row height when this is selected, heights of table rows are automatically adjusted to contents of table rows.



17.11. List of commands available in the printout module

This module enables printouts. The following commands are available:

RBCT ACTDOC

Available for a selected document located in the option tree; activates a selected document (for its edition) in the edition layout. There may be many documents created, but only one is active, i.e. may be edited.

RBCT ADDTOPRINT

Available for a selected view located in the position tree; composes a final printout. The layout, to which the view is to be added, must be active. The command adds a selected view to the current layout (printout); NOTE: a view may be contained only on one printout.

RBCT_ADDALLTOPRINT

Available for a selected document provided in the position tree; composes a final printout. The layout, to which the view is to be added, must be active. The command adds all the views belonging to a given document (not added to other printout, yet) to the current layout.

RBCT FITVIEWS

Available for an active document – provides greater work convenience to the user. The edition layout must be active. The command results in adjusting dimensions of views included in the edition layout to the current size of the AutoCAD® program window.

RBCT DELPRINTOUT

Available for a selected printout in the printout tree; the command is used to delete a printout. The command also deletes the corresponding layout.

RBCT REMFROMRINT

Available for a selected view provided in the printout tree; the command removes a view from a printout; the command removes the view only from a printout - it remains in a document and may be reused (e.g. by adding it to other printout).

RBCT DELALLFROMPRINT

Available for a selected document located in the printout tree; the command removes views belonging to a selected document from the printout. The views remain in a document and may be reused (e.g. by adding them to other printout).

RBCT DELDOC

Available for a selected document located in the position tree; the command deletes a document from the list of documents created for a given position.

RBCT EDITVIEW

Available for a selected view provided in the printout tree; the command switches to the edition mode for a selected view. The command enables you to switch from edition of a printout to edition of a view (drawing) included in it. The command activates the edition layout and a document whose component is the selected view.

RBCT EDITDOC

Available for a selected document provided in the printout tree; the command used to switch to the edition mode of a selected document. The command enables you to switch from edition of a printout to edition of views (drawings) included in it. The command activates the edition layout and the selected document.

RBCT MEDIT ON

Available for an active document containing one view; the command allows editing a document in the model layout.

RBCT MEDIT OFF

The command restores the standard functionality of the model layout.

RBCT REG LAYER

Available for the active document and view; after adding a layer, you can change its name by means of the options available in the AutoCAD® program. The program manages layers within the available views and documents. A layer must be added using the **AutoCAD® Structural Detailing** options (the problem concerns only the layers used in the edition layout).

RBCT DELETEPOS

Available for a position selected in the position tree; the command is used to delete a position; NOTE: only positions that are defined may be deleted - positions created on the basis of model elements cannot be deleted.

RBCT RENAMEPOS

Available for a position selected in the position tree; the command enables changing the name of a position.

RBCT RENAMEVIEW

Available for a view selected in the position tree; the command enables changing the name of a view.

RBCT RENAMEDOC

Available for a document selected in the position tree; the command enables changing the name of a document.

RBCT RENAMEPRINTOUT

Available for a printout selected in the printout tree; the command enables changing the name of a printout; a printout name is always identical to the name of a layout (printout) corresponding to it.

RBCT ACTVIEW

Available for a view selected in the position tree; the command makes the view of the AutoCAD® program which corresponds to the selected view, become active; NOTE: a view must be a component of the active document.

RBCT_ADDTEMPLATE

The command results in adding a new template in the dialog; which opens once this option is selected. You can specify the name of a new template and choose one of the registered template types.

RBCT DELTEMPLATE

Available for a template selected in the template tree; the command deletes a selected template.

RBCT RENAMETEMPLATE

Available for a template selected in the template tree; the command allows changing a name of a selected template.

RBCT ACTTEMPLATE

Available for a template selected in the template tree; the command activates a selected template in the template layout (it enables its edition).

RBCT ADDVIEWPORT

Accessible for an active template in the template layout; the command adds a view to the active template in the dialog box; which opens once this option is selected. You choose view name, scale and type; name and scale of the view may be changed at any time, as regards the type, it cannot be modified.

RBCT DELVIEWPORT

Accessible for a view selected in the template tree; the command enables deleting a view.

RBCT APPENDDOC

Available for a position selected in the position tree; the command adds a document to the selected position in the dialog; which opens once this option is selected. You can specify the name of a document and select a template based on which the document is to be created.

RBCT CNGSCALE

Accessible for the active view located in the edition layout; the command allows changing the view scale; the scale is expressed as the natural number n, which denotes the scale 1 : n.

RBCT SETVIEWRANGE

Available for the active view located in the edition layout; the command allows setting the view area visible on the printout. You select (with a rectangle) a part of the view which is to be visible on the printout. Edit operations in the edition layout do not result in modification of a defined print area.

RBCT REFRESHDOC

Accessible for a document selected in the position tree; the command enables refreshing a selected document. If the element of a structure model (for which the position has been defined), changes, then the documents created for this position are refreshed. A document that needs to be refreshed is marked with a red diagonal.

RBCT REFRESHALLDOC

Available for a position selected in the position tree; the command enables refreshing all the documents belonging to a selected position. If the element of a structure model (for which the position has been defined), changes, then the documents created for this position are refreshed. A document that needs to be refreshed is marked with a red diagonal.

RBCT PRINTVFRAMEON

Available for all defined, printout views.

RBCT PRINTVFRAMEOFF

Available for all defined, printout views; areas marked with rectangles (they determine views of the AutoCAD® program) will not be printed.

RBCT ADDDETAILVIEW

Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view. Once the command is selected, you select (with a rectangle) a part of the active view that is to become a new view. A view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout. A new view is added to the position tree; the new view contains copies of the elements included in the selected rectangle. The created view may be edited independently of the initial view. The command is useful when it is necessary to have another drawing (of the same part) to edit it independently or two drawings of different scales.

RBCT_ADDVIEW

Accessible for the active view provided in the edition layout; the command enables adding a new view based on the active view. Once the command is selected, you can select (with a rectangle) a part of the active view that is to become a new view. A view created in this manner is of the same type as the initial view and is positioned in the same place in the edition layout. A new view is added to the position tree; the new view contains the same elements as those included in the selected rectangle. If the created view is to be edited; it also makes changes in the initial view. The command is useful when several independent views are to be created out of one large view (drawing) which is not added to the printout; all the views created are assigned the same scale.

RBCT DELVIEW

Available for a view selected in the position tree; the command enables deleting a view that has been added by means of the following commands: RBCT_ADDDETAILVIEW or RBCT_ADDVIEW.

NOTE: The original view contained in a template, from which the document has originated, cannot be deleted.

RBCT_SHOWIEW

Accessible for a view selected in the position tree; the command allows edition in the active view; if additional views have been added to the document during edition, then not all the views are visible in the edition template.

RBCT REGMODELLAYERS

Available for the whole project; the command enables work optimization; once this command is selected, you can declare the layers intended for work in the model; while working in the edition layout. Layout (printout) or template layout, the layers are frozen. It brings about optimization of the **REGENALL** command operation.

18. AUTOMATIC DRAWING GENERATION

18.1. Automatic drawing generation

After you define a steel structure model, use this option to generate workshop drawings for selected positions. After choosing a position, you can automatically generate drawings for single parts or assemblies.

To begin generating drawings, open the Automatic drawing generation dialog from:

- Menu: Steel / Automatic printouts
- Ribbon: ASD Drawings / Generate drawings / Automatic printouts
- Toolbar: Printout / Automatic printouts
- Command line: RBCS_AUTOPRINT.

To generate final drawings:

- 1. Select positions for which drawings are to be generated.
- 2. Select templates for drawings of single parts.
- 3. Select templates for drawings of assemblies.
- 4. Select formats and scales.
- 5. Determine rules of assigning names to files.
- 6. Run automatic generation of final drawings.
- 7. Generate a final table.

After you select structure elements, use the tabs on the Automatic drawing generation dialog to specify parameters:

- Templates
- Formats and scales
- Options
- Bill of materials

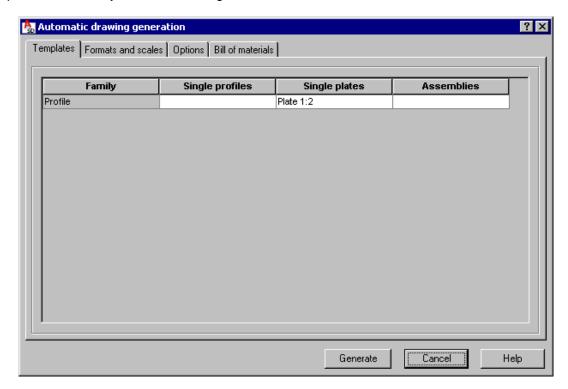
Click OK to start the operation of automatic drawing generation.

See also:

Example of positioning and automatic generation of printouts

18.2.Templates

On the Templates tab of the Automatic drawing generation dialog, you can define the type of a part and the family to which it belongs.



Assign a single part template or an assembly template to each family:

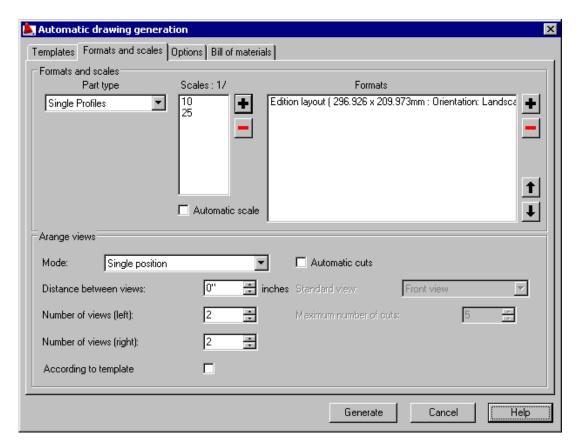
- For profiles or bent profiles, assign a default single part template.
- For plates, assign a default single part template.
- Assign a default assembly template.

AutoCAD®Structural Detailing templates define a set of parameters for each view and for a set of views and scales.

For automatic generation of drawings, a scale for views is defined in the dialog rather than taken from a template.

18.3. Formats and scales

On the Formats and scales tab of the Automatic drawing generation dialog, you can define scales, formats, and views for an option (object) selected in the Part type field.



For Scales, you can define scale values between 0.1 and 500. To add a scale value, click the + button. Click the – button to delete the currently selected scale value from the list. If you select Automatic scale, the Scales field is no longer available, and the scale is calculated by the software in such a way that it lays out the drawings on the printout optimally. For Formats, you can specify any number of template files. To add a template to the list, click the + button (an additional dialog opens, where you select a *.dwt format file). To delete the currently selected template from the list, click the − button. After you choose a template on the list, you can click the ↑ ↓ buttons to move the template up or down in the list. When you generate views of parts, templates are viewed from top to bottom of the list.



For templates used for automatic printouts, it is necessary to specify the Layout option in the definition of the plot area in the plot settings.

Under Arrange views, specify view parameters:

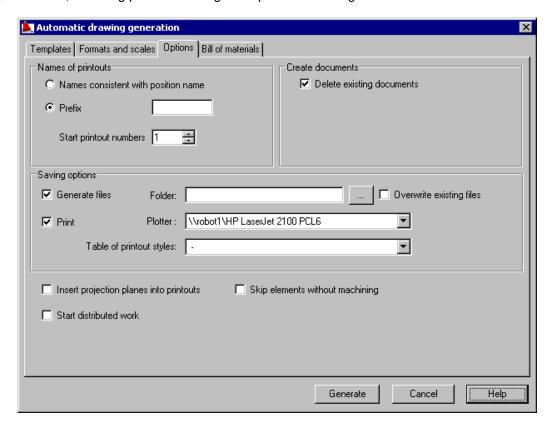
- Distance between views distance between views as they are displayed on the printout
- Number of views (left) and number of views (right) number of views (sections) located in the drawing to the left or to the right of the front view.
- Mode
 - Single position –select this to place each position in a separate drawing.
 - Assembly and parts select this to display all components of an assembly in one printout.
 - Many positions after you select a format, positions are arranged in drawings so that each drawing contains many positions and all the positions are included in the drawings.
 - Templates of plate marking (T-line) in this mode, plate drawings will be made in the scale 1:1 and will be arranged so that the maximum use of drawing space is ensured (positions not corresponding to plates are ignored in this work mode).

After selecting the *According to template* option, the previous options become unavailable. Views are generated based on the template settings.

If the *Automatic cuts* option is selected, cuts are created in views based on the user-specified parameters (standard view and maximum number of cuts).

18.4.Options

On the Options tab of the Automatic drawing generation dialog, you can specify various parameters, including printout naming and options for saving.



Under Names of printouts, you can select a method of defining names of files and layouts. A printout name may be identical to the position name or you can specify a prefix for the printout name. Also, you can specify a page number from which the printout numbering will start.

Under Create documents, If Delete existing documents is selected, all documents with the specified name included in the existing printouts will be deleted when drawings are generated.

Under Saving options, specify parameters:

- Selecting Generate files enables selection of a folder where DWG files containing single printouts will be located (if Overwrite existing files is selected, all existing files will be replaced when drawings are generated).
- Select Print to output the file to a plotter.
- Select an available style from the Table of printout styles.

If Overwrite existing files without confirmation is selected, files located in a selected folder will be replaced with newly generated files.

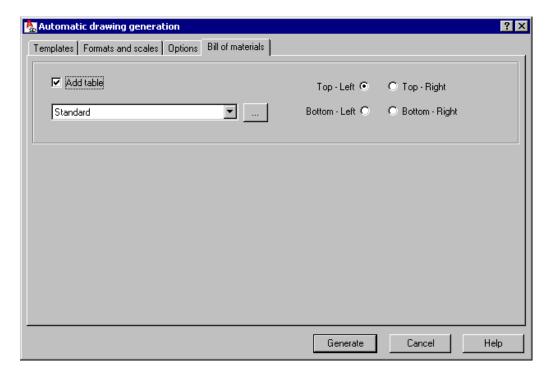
If you select Insert projection planes into printouts, viewports will be inserted into printouts. If the option is not selected, blocks will be inserted into printouts.

Select Start distributed work in order to divide a project into several parts (distributed work on a document); it ensures reduction of size of a file containing project data (quicker loading of a *.dwg file).

If the Skip elements without machining option is selected, elements without machining are not presented on printouts.

18.5.Bill of materials

On the Bill of materials tab of the Automatic drawing generation dialog, you can specify parameters for a bill of materials table.



If Add table is selected, a steel summary table will be added to the printout. Specify the corner of the printout in which to locate the table:

- Top Right
- Top Left
- Bottom Right
- Bottom Left

Click the Browse (...) button to open an additional dialog where you can choose a template of a steel summary table.

18.6. Printout templates

Printouts in *AutoCAD® Structural Detailing - Steel* (printout layouts) can be created based on user-created DWT files which are adjusted to typical formats and include templates of borders and drawing data tables according to user standards. Drawing info tables can be filled automatically with appropriate information specified in the project preferences on the Project Info tab. To do this, provide the following text variables on the Printout layout in appropriate places:

var office - Office field - Investor field var investor var_proj_name - Project name field - Address field var address var designer - Designer field - Verified field var verif - Date field var date - Revision field var rev var_note - Note field.

For the 'automatic printouts' mode you can also use the following variables associated with the printout:

var_pos - Position name

var scale - Drawing scale

var_part - Name of a part

var_material - Material of a part

var_length - Length of a part

var_weight - Weight of a part

var_paintarea - Painting area

var_quantity - Number of a given type of items in the project

var lengthnu - Length of a part without specified units

var_prndate - Date of generating the printout

var prndate - Time of generating the printout

var_namenu - Element name without specified units for plates

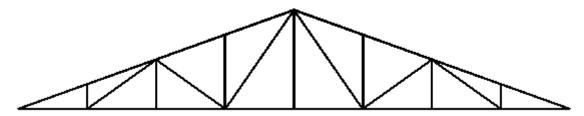
var family - Element family

var_surf - Surface finishing

var_piecexp - Specifies if a given part is included in an assembly.

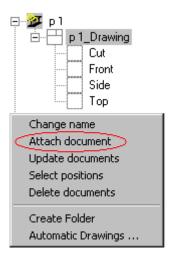
18.7. Example of creating drawings (printouts)

The example below is for creating drawings of a simple truss.

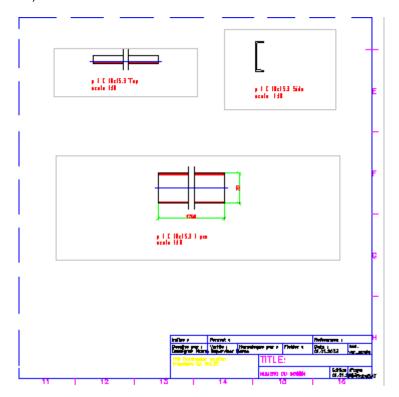


To create a drawing of structure elements:

- 1. On the Model tab of the Object Inspector dialog, select all the truss profiles (the profiles will be highlighted).
- 2. Right-click, and click Auto positioning.
- 3. In the Automatic positioning dialog:
 - On the General tab
 - o For Positioning level, select single part.
 - o For Prefix, select By family.
 - o For Number, specify numerical format, start from 1, step = 1.
 - On the Additional tab, select all options.
 - On the Numbering tab
 - For Sort by, select element shape.
 - o For Sorting criteria, select family.
- 4. Click Run; all the truss elements will be assigned positions according to the parameters defined in the Automatic positioning dialog.
- 5. On the Positions tab of the Object inspector dialog, select the first position p1 (the position will be highlighted).
- 6. Right-click, and click Attach document (see the drawing below).



- 7. In the Select template dialog, select the template Profile 1:10 that will be used to create documents (projections), and click OK. In the edition layout, the software has generated projections / views for the selected position.
- 8. On the Printouts tab of the Object Inspector dialog, right-click A1 ASD, and click Activate.
- 9. On the Positions tab, right-click the first position, and click Add to current Printout.
- 10. In the drawing area, click to specify the location of the document (projections of elements) in the printout layout. The selected drawing of the position has been added to A1 ASD on the Printouts tab in the Object Inspector dialog (part of this drawing is shown below).



18.8. Engineering drawing

18.8.1. Engineering drawing

Use this option to define and modify structure schemes. Access the option using one of the following methods from:

- Menu: Steel / Engineering drawings / Engineering drawings
- Ribbon: ASD Model / Groups / Engineering drawings
- Toolbar: Engineering drawings / Engineering drawings
- Command line: RBCS SCHEME.

After selecting the option, select elements belonging to the scheme.

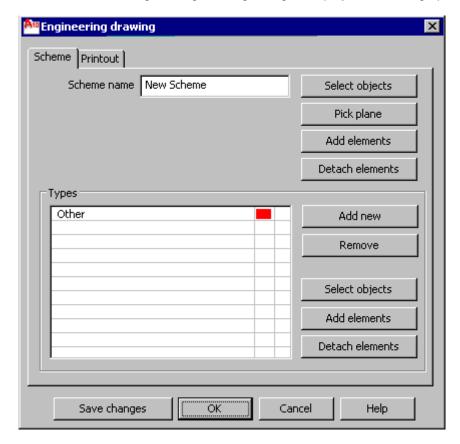
After selecting the structure elements, the *Engineering drawing* dialog displays; it has the following tabs:

- Scheme
- Printout.

Click **Save changes** to save changes in a defined scheme without closing this dialog. Click **OK** to save changes in a defined scheme and close this dialog.

18.8.2. Scheme

Select the Scheme tab in the Engineering drawing dialog to display the following options.



The Scheme tab has the following options:

 Scheme name - use it to specify a scheme name; after defining or changing the name, click Save changes

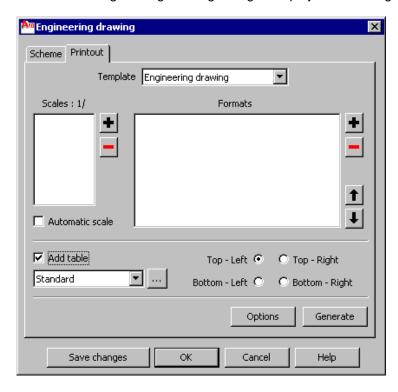
- Select objects clicking this button defines or modifies a selection of elements for a scheme; click this button to display the whole structure where you can select elements of the scheme; selected elements are displayed in red
- Pick plane clicking this button defines or modifies the XY plane of the scheme; click
 this button, next select the main part (it aligns the coordinate system) using one of the
 options:
 - *UCS* the XY plane is set with respect to the current UCS *WCS* the XY plane is set with respect to WCS 3 *points* the XY plane is defined by selecting 3 points
- Add elements clicking this button adds elements to the current element selection; click
 this button to display the whole structure where you can select elements to be added to
 the scheme; selected elements are displayed in red
- **Detach elements** clicking this button removes elements from the current element selection; click this button to display only elements belonging to the scheme; then select elements to be removed from the scheme.

Use the options in the *Types* field to define or modify subgroups in a selected scheme:

- Add new clicking this button defines new types; click Add new to define a new type with a default name (which can be changed)
- Remove clicking this button removes a selected type from the list
- Select objects clicking this button defines or modifies a set of elements that belong to
 a selected type; click this button to display only elements belonging to the scheme; then
 select elements belonging to a type (available types are: selection, family, position or
 mark)
- Add elements clicking this button adds elements to a selected type; click this button to display only elements belonging to the scheme; then select elements to be added to the type
- Detach elements clicking this button removes elements from a selected type; click this button to display only elements belonging to the scheme; then select elements to be removed from the type.

18.8.3. **Printout**

Select the *Printout* tab in the Engineering drawing dialog to display the following options.



At the top of the dialog select a template; a printout is generated based on this template. You can define any number of values in the *Scales* field; the allowable scale values are those from the interval 0.1 and 500. Click + to define a new scale value; click – to remove a selected scale value from the list. Under this field is the Automatic scale options; if selected, the Scales 1:n field is not available and the software calculates the scale as to ensure that the printout layout if optimally filled with drawings.

You can define any number of template files in the *Formats* field. Click + to add a new template to the list (it opens an additional dialog where you can select a *.dwt file); click - to remove a selected template from the list. To move a template up or down on the list, select a template and click \uparrow or \downarrow , respectively. When views of parts are created, templates are viewed from top of the list to the bottom.

If the Add table option is selected, a steel summary table is added to the printout; to specify the position of the table on the printout, select one of the following options:

- Top Right (corner of the printout)
- Top Left
- Bottom Right
- Bottom Left.

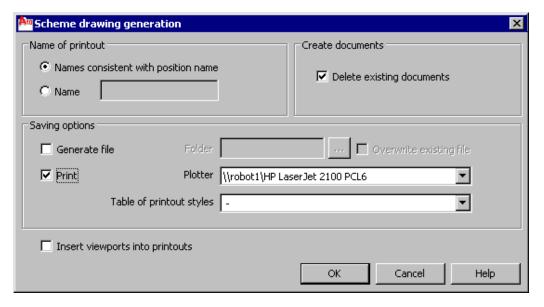
Click (...) to open the **Steel summary tables – style manager** dialog where you can select a steel table template.

Click **Options** to open a dialog with options used for generating a scheme drawing.

Click **Generate** to start generating a scheme drawing.

18.8.4. Generating a scheme drawing

Click **Options** in the Engineering drawing dialog to display the following dialog.



The Name of printout field allows you specify the way of defining names of files and tabs; you can define a printout name identical to the position name or specify a prefix for a printout name. Below specify a number from which to start numbering the printout pages.

If the Delete existing documents option is selected, all documents with the specified name are deleted from the existing printouts when generating drawings.

Specify the following parameters in the Saving options field:

- Folder select this option to select a folder where DWG files with single printouts are located (if Overwrite existing file is selected, all existing files are replaced when generating drawings)
- Print select this option to select a plotter file type
- the selection list includes available printout styles.

After selecting Insert viewports *into printouts*, viewports are inserted on printouts; if this option is cleared, blocks are inserted on printouts.

19. LINK WITH OTHER PROGRAMS

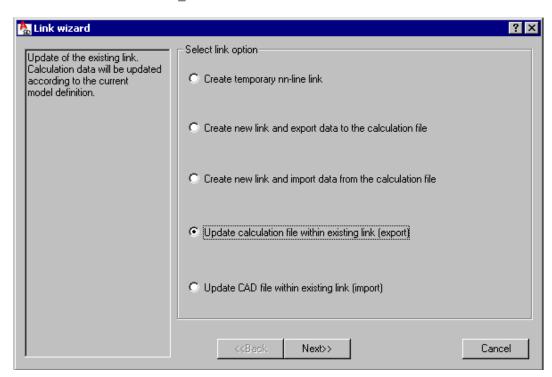
19.1.AutoCAD Structural Detailing - Robot link

The link between Autodesk Robot Structural Analysis (a calculation program) and AutoCAD® Structural Detailing (used to create drawings) has been designed in order to enable you to:

- Perform calculations of a defined CAD 3D model using Autodesk Robot Structural Analysis at any work stage. Calculations may be performed both for a whole structure and for a selected structure part.
- Apply a calculation model created in Autodesk Robot Structural Analysis as the basis for creation of a detailed CAD model and documentation of a designed structure.
- Perform full synchronization (from Autodesk Robot Structural Analysis to AutoCAD® Structural Detailing and vice versa) of changes made in a structure calculation model as well as in its CAD model.

Updating the CAD file and the file containing the structure calculation model is carried out in the Link Wizard dialog shown below. To open the Link wizard dialog from:

- Menu: Steel / Analysis Interface Robot
- Ribbon: ASD Model / Tools / Analysis Interface Robot
- Toolbar: General / Analysis Interface
- Command line: RBCS R2R.



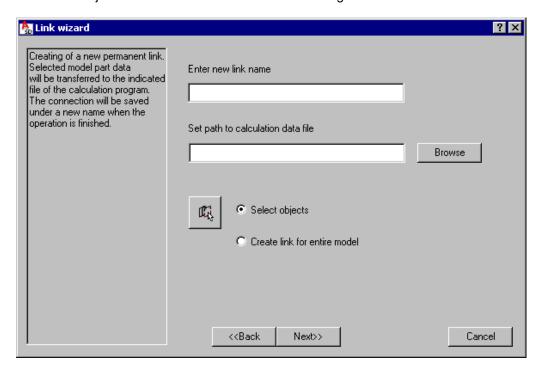
To begin updating data for a selected Autodesk Robot Structural Analysis - AutoCAD® Structural Detailing connection defined in the current CAD file, choose an available operation:

- To create a new link (permanent or temporary), select one of the first three options.
- To update an existing link, select one of the last two options (update of a structure calculation model based on changes made in AutoCAD® Structural Detailing or vice versa).

The left pane of the Link wizard dialog displays a short description of the option selected on the right.

After you click Next, the Link wizard dialog changes as shown below, allowing you to:

- Assign a name of the defined link or select a name of the existing link
- Create or select a file with an RTD extension (a file of the Autodesk Robot Structural Analysis calculation program) in which calculation results will be saved
- Select objects that will be considered while creating a link.



To create a new link:

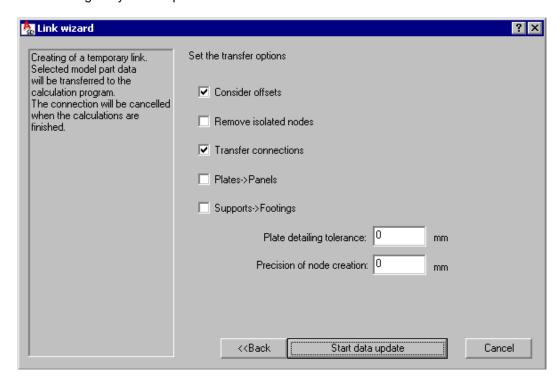
- For Enter new link name, specify a link name. This field is available only when you're
 creating a permanent link. The link is saved under a specified name in the current
 DWG file. Because the name identifies links, it must be unique in a given file.
- For Set path to calculation data file, specify a calculation file with an RTD extension (an Autodesk Robot Structural Analysis file) to which calculation results for the created link will be saved. This field is available only when you're creating a new permanent link. As data is updated, the path is displayed in the field, but is read-only. You can specify a file by clicking Browse, and then using one of the standard Windows dialogs that displays:
 - Open dialog displays when you are importing data from an existing file that contains a calculation model (RTD file).
 - Save dialog displays when you are exporting data to a new file that is being created at this stage.
- The options provided at the bottom of the Link wizard dialog let you select objects that are to be link components (the options are available only when a new link is being created):
 - o Create link for entire model select this option in order to use the entire structure
 - Select objects select this option in order to specify individual structure objects.
 When you use this option, you can click the button that then lets you select objects in the drawing.

To update an existing link:

- For Choose link, select one of the permanent links available in the current DWG file.
 After a link is selected, it becomes active. The list of all permanent links defined in the current DWG file may be viewed (and configured, if needed) in the additional dialog that opens after you click See list.
- Click Next to display a dialog (shown below) where you can specify parameters of data transfer. The options in this dialog depend on the option (direction of data update) chosen in the first dialog.

- For a new link, the default options are selected.
- For an existing link, the options that are selected are those that were applied for the selected link during the last operation of data update.

You can change any of the options.



To create a link and begin data update based on the options selected in the above dialogs, clickStart data update. The current dialog is closed, and a data update report displays. Data update is an obligatory operation while creating a new link.

This dialog displays the run of the data update process. A progress bar displays at the bottom of the dialog, reporting the progress of the data update operation. Messages related to the data update process display also:

- Date of the update process
- File names (source file and target file)
- Direction of the data update process (from AutoCAD® Structural Detailing or to AutoCAD® Structural Detailing)
- Warnings

To save the contents of the data update report in a text file, click Save log file. A text file may be corrected manually by the user.

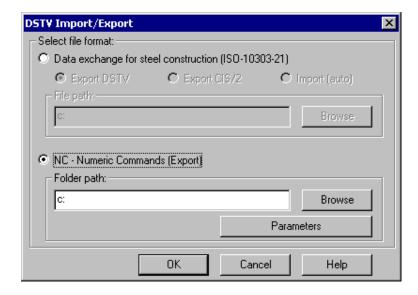
The OK and Save log file buttons are available only after the data update process is completed.

19.2.DSTV import/export

Use this option to read or save a DSTV format file.

To begin working with a DSTV file, open the DSTV Import/Export dialog from:

- Menu: Steel / DSTV data export
- Ribbon: ASD Model / Tools / DSTV data export
- Toolbar: General / DSTV data export
- Command line: RBCS_DSTV.

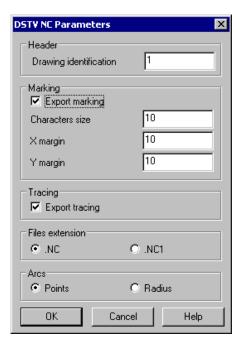


Available methods of importing or exporting DSTV or CIS/2 format file:

- A file containing description of a steel structure model (a file that is compliant with the ISO-10303-21 standard) can be both imported and exported. For File path, specify the file location on disk (file location can also be specified after clicking Browse). This import/export mode results in generation of one file with *.stp extension including description of a structure model.
- Files containing descriptions of steel structure parts to be applied when numeric commands are used (the files can only be exported). For Directory path, specify location of the folder in which files with description of structure parts will be saved (folder location can also be indicated after clicking Browse). In this mode (for each part of a steel structure that has been assigned a position), a separate file containing description of parts is created in the indicated folder. Click Parameters in order to open the DSTV NC Parameters dialog.

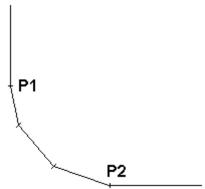
19.3.DSTV NC Parameters

Use the DSTV NC Parameters dialog to define additional parameters of pointing and marking. The dialog opens after you click Parameters in the DSTV Import / Export dialog.

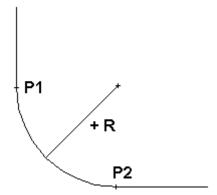


In descriptions of steel structure parts, used in the numeric control of machine tools, you can provide the following parameters:

- Header a drawing will be identified by means of a specified text / number
- Marking if Export marking is selected, the marking block will be included in the exported file. Specify parameters of marking a steel element
 - Size of characters
 - o Position of the point where a description will be inserted (X margin and Y margin)
- Tracing if this is selected, the pointing block will be included in the exported file.
- Files extension (NC or NC1)
- Method of arc definition:
 - Points by means of points positioned on the arc. The arc will be approximated between the beginning point P1 and the end point P2 by segments between points on the arc.



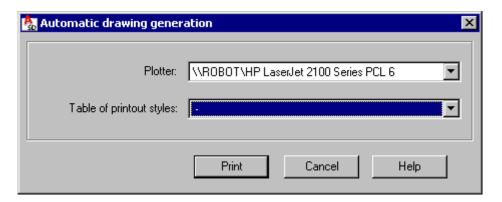
 Arc - by means of the arc radius. Between the beginning point P1 and the end point P2, the arc will be defined based on the radius value.



20. AUTOMATIC DRAWING GENERATION - SAVING TO PLOTTER

Use this option to automatically generate workshop plotter drawings. To access the option, select from:

- Ribbon: ASD Drawings / Generate Drawings / Automatic plotting
- Toolbar: Printouts / Automatic plotting
- Command line: AUTOPLOT.



Specify plotting parameters:

- For Plotter, choose a type of plotter file.
- For Table of printout styles, select from the list of available styles.

Click Print to start generation of drawings by a selected plotter type.