

Autodesk Navisworks Manage 2012

User Guide

The Autodesk logo is displayed in white text on a black rectangular background. The word "Autodesk" is written in a sans-serif font, with a registered trademark symbol (®) at the end. The logo is oriented vertically, reading from bottom to top.

April 2011

©2011 Autodesk, Inc. All Rights Reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 3DEC (design/logo), 3December, 3December.com, 3ds Max, Algor, Alias, Alias (swirl design/logo), AliasStudio, Alias|Wavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, Autodesk Intent, Autodesk Inventor, Autodesk Map, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backburner, Backdraft, Built with ObjectARX (logo), Burn, Buzzsaw, CAiCE, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design Web Format, Discreet, DWF, DWG, DWG (logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DXF, Ecotect, Exposure, Extending the Design Team, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, FMDesktop, Freewheel, GDX Driver, Green Building Studio, Heads-up Design, Heidi, HumanIK, IDEA Server, i-drop, ImageModeler, iMOUT, Incinerator, Inferno, Inventor, Inventor LT, Kaydara, Kaydara (design/logo), Kynapse, Kynogon, LandXplorer, Lustre, MatchMover, Maya, Mechanical Desktop, Moldflow, Moonbox, MotionBuilder, Movimento, MPA, MPA (design/logo), Moldflow Plastics Advisers, MPI, Moldflow Plastics Insight, MPX, MPX (design/logo), Moldflow Plastics Xpert, Mudbox, Multi-Master Editing, Navisworks, ObjectARX, ObjectDBX, Open Reality, Opticore, Opticore Opus, Pipeplus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, ProMaterials, RasterDWG, RealDWG, Real-time Roto, Recognize, Render Queue, Retimer, Reveal, Revit, Showcase, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), Sparks, SteeringWheels, Stitcher, Stone, StudioTools, ToolClip, Topobase, Toxik, TrustedDWG, ViewCube, Visual, Visual LISP, Volo, Vtour, Wire, Wiretap, WiretapCentral, XSI, and XSI (design/logo).

LightWorks, the LightWorks logo, LWA and LWA-Enabled are registered trademarks of LightWork Design Ltd. The LWA-Enabled logo, Interactive Image Regeneration, IIR, A-Cubed, Feature-Following Anti-Aliasing and FFAA are all trademarks of LightWork Design Ltd. All other trademarks, images and logos remain the property of their respective owners. Copyright of LightWork Design Ltd. 1990-2007, 2008.

This software is based in part on the work of the Independent JPEG Group.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

This User Guide was last updated on 11 April 2011.

Contents

Chapter 1	Welcome to Autodesk Navisworks Manage 2012	1
	What Is New in This Release?	1
	How to Get Assistance	11
	Find Information Using InfoCenter	11
	Overview of InfoCenter	11
	Search for Information	12
	Access Subscription Center	13
	Use Communication Center	15
	Save and Access Favorite Topics	17
	Use the Help System	18
	Specify InfoCenter Settings	23
	Get More Help	25
	Learn the Product	26
	View the Product Readme	27
	Join the Customer Involvement Program	28
Chapter 2	Installation	29
	Quick Start to Stand-Alone Installation	29
	Prepare for Installation	29
	System Requirements for Stand-Alone Installation	30
	Install Microsoft .Net Framework 4.0	31
	Understand Administrative Permission Requirements	32

Locate Your Autodesk Navisworks Serial Number and Product Key	32
Avoid Data Loss During Installation	33
Choose a Language	33
Configuration	34
Install Multiple or Bundled Products	36
Install and Run Autodesk Navisworks Manage 2012	36
Install Autodesk Navisworks	37
Launch Autodesk Navisworks	39
How to Launch Autodesk Navisworks in Another Language	40
Add or Remove Features	41
Repair Autodesk Navisworks Manage 2012	42
Uninstall Autodesk Navisworks Manage 2012	42
Move to Autodesk Navisworks from a Previous Release	43
Install Autodesk Navisworks for Multiple Users	44
Quick Start to Network Administration and Deployment	44
Deployment Preparation	44
Set Up Network Tools and Your License Server	50
Distribute the Program	52
Distribute an Autodesk Navisworks Product	54
Set Up a Deployment	54
Preliminary Tasks for a Network Deployment	55
Choose a Language	56
Your Deployment Choices	57
Create a Deployment	65
Modify a Deployment (optional)	69
Point Users to the Administrative Image	70
Uninstall an Autodesk Product	70
Installation Troubleshooting	71
General Installation Issues	71
How can I check my graphics card driver to see if it needs to be updated?	71
How do I switch my license from stand-alone to network or network to stand-alone?	72
When performing a Typical installation, what gets installed?	72
Why should I specify the Project Folder and Site Folder?	73
How do I share the Autodesk Navisworks settings on a site and project basis?	73
How do I change which exporter plugins are installed?	75
How do I register and activate Autodesk Navisworks?	75
When should I reinstall the product instead of repairing it?	76

When I uninstall my software, what files are left on my system?	77
Deployment Issues	77
Is there a checklist I can refer to when performing a deployment?	77
Where should deployments be located?	77
Where can I check if service packs are available for my software?	78
How do I choose between 32-bit and 64-bit deployments?	78
What are information channels?	78
What are additional deployment configuration options?	79
Licensing Issues	80
What is the difference between a stand-alone license and a network license?	80
What is the benefit to using a network licensed version of the software?	81
What is Internet Explorer used for?	81
Networking Issues	81
Where do I find my server name?	81
If I choose to create a log file, what kind of information does the log file contain?	82
What is an administrative image (MSI) file?	82
What is the impact of selecting all products to be included in the administrative image?	82
How should I configure a network license server for a firewall	83
Uninstall and Maintenance Issues	83
When adding or removing features, how can I tell what features get installed by default?	83
Is it possible to change the installation folder when adding or removing features?	83
When should I reinstall the product instead of a repair?	84
When I uninstall my software, what files are left on my system?	84
Chapter 3 Quick Start	85
Start and Quit Autodesk Navisworks	85
Automatically Save and Recover Autodesk Navisworks Files	86
Command Line Options	88
The User Interface	91
Parts of Autodesk Navisworks Interface	91
Application Button and Menu	93
Quick Access Toolbar	96

	Ribbon	98
	Tooltips	107
	Keytips	107
	Navigation Tools	108
	The Classic User Interface	108
	Scene View	129
	Dockable Windows	132
	Status Bar	138
	Undo/Redo Commands	139
	Autodesk Navisworks Workspaces	140
	Default Keyboard Shortcuts	142
	Navigation with the Wheel Button	147
	Autodesk Navisworks Options	150
	Location Options	154
	Graphics System	155
	Display Units	157
	Profiles	158
	Search Directories	159
	Gizmos	160
Chapter 4	Work with Files	163
	Native File Formats	163
	Compatible CAD Applications	164
	Supported CAD File Formats	168
	Supported Laser Scan File Formats	169
	Use File Readers	170
	3DS File Reader	170
	ASCII Laser Scan File Reader	171
	Bentley AutoPLANT File Reader	172
	CIS/2 File Reader	172
	DWG/DXF File Reader	175
	Overview of Object Enablers	177
	DWF/DWFX File Reader	179
	DGN File Reader	180
	Faro Scan File Reader	181
	FBX File Reader	181
	IFC File Reader	182
	IGES File Reader	183
	Inventor File Reader	183
	JTOpen File Reader	184
	Leica Scan File Reader	185
	MAN File Reader	185
	Parasolid File Reader	187
	PDS File Reader	187
	Pro/ENGINEER File Reader	187
	Riegl Scan File Reader	188

RVM File Reader	188
SAT File Reader	189
SketchUp SKP File Reader	189
STEP File Reader	190
STL File Reader	190
VRML File Reader	191
Z+F Scan File Reader	192
Use File Exporters	192
AutoCAD File Exporter	192
Add the ARX Plugin	193
Use the ARX Plugin	194
CAD Preview	195
Revit File Exporter	200
MicroStation File Exporter	201
Load the MDL Plugin	202
Export Files from the Key-In Command Line	202
Export Files from the Command Line	203
Customize the DGN File Exporter Options	204
Viz and Max File Exporter	205
ArchiCAD File Exporter	206
Manage Files	209
Open Files	209
Create Files	210
Save and Rename Files	210
2D and Multi-Sheet Files	213
Add Sheets/Models to the Currently Opened File	214
Project Browser Window	214
Work with 2D and Multi-Sheet Files	218
Add Geometry and Metadata to the Current Sheet/Model	219
Complex Datasets	221
Append Geometry and Metadata to the Current Scene	222
Delete Files	222
Adjust Units and Transform	223
Refresh Files	225
Merge Files	225
Email Files	227
Receive Files	227
Batch Utility	228
Use Batch Utility	228
Command Line Options for Batch Utility	236
View Scene Statistics	238
Chapter 5 Explore Your Model	239
Navigate a Scene	239

Orientation in a 3D Workspace	240
Product-Specific Navigation Tools	241
Navigation Bar Tools	242
SteeringWheels Tools	247
Classic Navigation Modes and Tools	264
ViewCube	273
Overview of ViewCube	273
ViewCube Menu	276
Reorient the View of a Model with ViewCube	277
Set the View Projection Mode	281
Home View	282
Examine Individual Objects with ViewCube	283
Navigation Bar	283
Overview of Navigation Bar	284
Reposition and Reorient the Navigation Bar	285
Control the Display of Navigation Tools on the Navigation Bar	286
SteeringWheels	287
Overview of SteeringWheels	287
Wheel Menu	291
View Object Wheels	292
Tour Building Wheels	294
Full Navigation Wheels	296
2D Navigation Wheel	298
3Dconnexion 3D Mouse	298
Camera	301
Set Camera Projection	301
Control the Field of View	302
Position and Focus Camera	302
Navigation Aids	306
Head-Up Display	306
Reference Views	307
Focus	310
Hold	310
Control the Realism of Your Navigation	311
Gravity	311
Crouching	311
Collision	312
Third Person View	313
Chapter 6 Control Model Appearance and Render Quality	317
Control Model Appearance	317
Select Render Mode	317
Full Render	318
Shaded	318
Wireframe	319

	Hidden Line	319
	Add Lighting	319
	Full Lights	320
	Scene Lights	320
	Head Light	321
	No Lights	322
	Select Background Effect	323
	Adjust Displaying of Primitives	325
	Surfaces	325
	Lines	325
	Points	326
	Snap Points	326
	Text	327
	Control Render Quality	327
	Use Culling	327
	Make Objects Required	330
	Control Rendering of Objects	331
	Adjust Scene Rendering During Navigation	331
	Accelerate Display Performance	333
	Adjust Presenter Materials	334
	Stereo Rendering	334
Chapter 7	Review Your Model	337
	Select Objects	337
	Interactive Geometry Selection	337
	Selection Tree Window	338
	Selection Tools	341
	Selection Commands	343
	Set Selection Resolution	345
	Set Highlighting Method	346
	Hide Objects	348
	Find Objects	349
	Find Items Window	349
	Quick Find	355
	Find All Sheets and Models Containing the Selected Object	356
	Find Items in Other Sheets and Models Window	357
	Create and Use Sets of Objects	360
	Sets Window	360
	Create and Manage Selection and Search Sets	362
	Compare Objects	365
	Object Properties	367
	Properties Window	367
	Custom Properties	369
	External Database Links	371
	Manipulate Object Attributes	382
	Transform Objects	383

Change Object Appearance	388
Snapping	389
Reset to Original Values	390
Measure Tools	391
Measure Tools Window	391
Measuring	393
Comments, Redlines, and Tags	399
Use Comments, Redlines, and Tags	399
Comments Window	400
Redline Tools Panel	403
View Redlines and Tags	410
Tags Panel	411
Edit Comments and Tags	412
Edit Redlines	413
Find Comments and Tags	414
Find Comments Window	414
Quick Find Comments	418
Find Tags	419
Manage Comment and Tag IDs	420
Links	421
Link Categories	421
Display Links	422
Customize Links	423
Add Links	427
Find and Follow Links	428
Manage Links	430
Quick Properties	432
SwitchBack	434
Appearance Profiler	437

Chapter 8 Use Viewpoints and Sectioning Modes 441

Create and Modify Viewpoints	441
Overview of Viewpoints	441
Saved Viewpoints Window	442
Save Viewpoints	447
Recall Viewpoints	448
Organize Viewpoints	448
Edit Viewpoints	449
Default Viewpoint Options	451
Share Viewpoints	454
Sectioning	455
Enable and Use Section Planes	456
Customize Section Plane Alignment	459
Move and Rotate Section Planes	461
Link Section Planes	464
Enable and Use Section Box	465

Chapter 9	Record and Play Animations	471
	Create and Edit Viewpoint Animations	471
	Play Animations and Scripts	475
	Share Animations	477
Chapter 10	Work Within a Team	479
	Collaborate Panel	479
	Collaboration Session	479
Chapter 11	Share Data	483
	Print	483
	Print Preview	483
	Print Setup	483
	Print Current Viewpoint	484
	Import Files	484
	Search Criteria Files	485
	Search Set Files	485
	PDS Display Set Files	486
	PDS Tag Files	487
	Viewpoints Files	488
	Clash Test Files	489
	Export Files	489
	3D DWF/DWFX Format	489
	Google Earth KML Format	490
	Autodesk FBX Format	492
	Export Images and Animations	494
	Export an Image	494
	Export a Rendered Image	496
	Export an Animation	496
	Piranesi EPix Format	497
	Current Search Criteria	498
	Search Set Files	498
	Viewpoints Files	498
	Viewpoints Report	499
	TimeLiner CSV	500
	Clash Test Files	500
	PDS Tag Files	500
Chapter 12	Animate Objects	501
	Overview of the Animator Tool	502
	Animator Window	502
	The Animator Toolbar	502
	The Animator Tree View	505
	The Animator Timeline View	508

The Manual Entry Bar	512
Scripter Window	513
The Scripter Tree View	513
The Events View	515
The Actions View	517
The Properties View	519
Create Object Animations	526
Work with Animation Scenes	527
Work with Animation Sets	529
Add Animation Sets	529
Update Animation Sets	530
Manipulate Geometry Objects	531
Work with Cameras	534
Work with Section Plane Sets	536
Work with Keyframes	538
Capture Keyframes	538
Edit Keyframes	538
Play Animation Scenes	539
Add Interactivity	540
Work with Animation Scripts	541
Work with Events	543
Work with Actions	545
Enable Scripting	547
Chapter 13 Create Photorealistic Visualizations	549
Overview of the Presenter Tool	549
Presenter Window	549
Use the Presenter Archives	551
Photo-Realistic Scene Rendering	553
Use Presenter Materials	556
Materials Tab	556
Apply and Remove Presenter Materials	556
Organize and Manage Materials	559
Edit Presenter Materials	561
Advanced Materials	565
Use Presenter Lights	568
Lighting Tab	568
Add and Position Lights	569
Organize and Manage Lights	571
Edit Lights	573
Shadow Casting	576
Advanced Lighting	577
Soft Shadows	578
Physically Accurate Lights	578
Volumetric Lights	579
Image-based Lighting	580

Use Presenter RPCs	583
RPC Tab	583
Use Presenter Rendering Effects	587
Effects Tab	587
Background Effects	588
Foreground Effects	593
Use Presenter Rendering Styles	594
Rendering Tab	594
Rendering Styles	595
Predefined Rendering Styles	596
Auto Exposure	598
Use Presenter Texture Space	598
Use Presenter Rules	600
Rules Tab	600
Predefined Rules	601
Custom Rules	602
The Presenter Rules Example	604
Chapter 14 Simulate Construction Scheduling	607
Overview of TimeLiner Tool	607
TimeLiner Window	608
Tasks Tab	609
Data Sources Tab	613
Configure Tab	615
Simulate Tab	617
Choose TimeLiner Columns Dialog Box	619
TimeLiner Rules Dialog Box	619
Field Selector Dialog Box	620
Refresh from Data Source Dialog Box	624
Simulation Settings Dialog Box	624
Overlay Text Dialog Box	630
Appearance Definitions Dialog Box	632
Get Started	633
TimeLiner Tasks	637
Create Tasks	639
Edit Tasks	641
Use Gantt Charts	643
Attach Tasks to Geometry	644
Attach Tasks Manually	645
Use Rules to Attach Tasks	646
Validate Project Schedule	649
Link to External Project Files	650
Supported Scheduling Software	651
CSV Support	654
Add and Manage Data Sources	654
Import Data from an External Project Schedule	654

Edit a Data Source	658
Delete a Data Source	658
Build Tasks from Data Sources	659
Synchronize Tasks with Project Changes	659
4D Simulation	660
Play Simulations	660
Configure Simulations	661
Simulation Playback	661
Simulation Appearance	661
Export	664
Add Animation	664
Overview	664
Add Animation to an Entire Schedule	665
Add Animation to Tasks	667
Add Scripts to Tasks	668
Chapter 15 Locate and Manage Interferences	669
Overview of Clash Detective Tool	669
Clash Detective Window	670
Batch Tab	672
Rules Tab	673
Select Tab	674
Results Tab	677
Report Tab	684
Set Up and Run a Clash Test	686
Clash Batches	686
Run Clash Tests	686
Manage Batches of Clash Tests	687
Merge Clash Tests from Multiple Files	687
Import Clash Tests	688
Export Clash Tests	689
Create Custom Clash Tests	690
Clash Rules	691
Select Items for Testing	696
Select Items for a Clash Test	696
Select Clash Test Options	697
Time-Based and Soft Clash Testing	698
Time-Based Clashing	699
Soft Clashing	700
Time-Based Soft Clashing	701
Run an Individual Clash Test	703
Clash Results	703
Understand Clash Results	704
Manage Clash Results	705
Review Clash Results	708
Visually Identify Clashes in a Model	708

	Add Review Comments and Redlines	713
	Time-Based and Soft Clash Test Results	714
	Report Clash Results	717
Chapter 16	Use the Autodesk Vault Add-In	723
	About the Autodesk Vault Add-In	723
	Launching the Vault Application	724
	Log into a Vault	724
	Log out of a Vault	725
	Understanding the Working Folder	725
	Check Out a File	727
	Get Files from a Vault	728
	Refresh a Vaulted File	729
	Check a File into a Vault	729
	Undo File Check Out	730
	Vault Settings	731
	Log In Dialog Box	731
	Check In Dialog Box	732
	Settings Dialog Box	733
	Select Vault Location Dialog Box	733
	Create Folder Dialog Box	734
	Select File Dialog Box	734
Chapter 17	Reference	737
	Animation Export Dialog Box	737
	Appearance Profiler Dialog Box	739
	Assign Clash Dialog Box	741
	Background Settings Dialog Box	741
	Collision Dialog Box	742
	Convert Object Properties Dialog Box	743
	Culling Options Dialog Box	743
	Customize Dialog Box	745
	Toolbars Tab	745
	Commands Tab	745
	Options Tab	746
	Default Collision Dialog Box	747
	Edit Key Frame Dialog Box	748
	Edit Link Dialog Box	751
	Edit Viewpoint Dialog Box	751
	Export Rendered Image Dialog Box	753
	File Options Dialog Box	754
	Culling Tab	755
	Orientation Tab	756
	Speed Tab	757
	Headlight Tab	757

Scene Lights Tab	758
DataTools Tab	758
Units and Transform Dialog Box	759
Image Export Dialog Box	760
InfoCenter Settings Dialog Box	761
General Node	761
Communication Center Node	762
Autodesk Channels Page	762
Balloon Notification Page	762
RSS Feeds Page	763
New Link Dialog Box	763
Options Editor Dialog Box	764
General Node	765
Undo Page	765
Locations Page	766
Environment Page	766
Auto-Save Page	766
Interface Node	767
Display Units Page	768
Selection Page	768
Measure Page	769
Snapping Page	770
Viewpoint Defaults Page	770
Links Page	771
Quick Properties Page	775
Developer Page	776
Display Page	777
3Dconnexion Page	781
Navigation Bar Page	782
ViewCube Page	783
SteeringWheels	784
User Interface Page	787
Model Node	787
Performance Page	787
NWD Page	790
NWC Page	790
File Readers Node	792
3DS Page	792
ASCII Laser Page	792
CIS/2 Page	793
DGN Page	793
DWF Page	796
DWG/DXF Page	797
Faro Page	801
FBX Page	801
IFC Page	802

Inventor Page	803
Leica Page	804
JTOpen Page	805
MAN Page	805
Parasolid Page	806
PDS Page	806
Riegl Page	807
RVM Page	808
SAT Page	809
SKP Page	810
STL Page	811
VRML Page	811
Z+F Page	812
File Exporters Node	813
DWG Page	813
Revit Page	817
DGN Page	819
Viz/Max Page	821
Tools Node	822
Clash Detective Page	822
TimeLiner Page	823
Presenter Page	824
Vault Page	826
Scripter Page	826
Animator Page	827
Publish Dialog Box	827
Piranesi EPix Dialog Box	829
QTVR Object Movie Settings Dialog Box	829
Section Plane Settings Dialog Box	830
Chapter 18 Glossary	833
Index	841

Welcome to Autodesk Navisworks Manage 2012



Autodesk Navisworks Manage 2012 software is a comprehensive project review solution for analysis, simulation, and communication of design intent and constructability. Multidisciplinary design data created in Building Information Modeling (BIM), digital prototype, and process plant design applications can be combined into a single integrated project model. Interference management and clash detection tools help design and construction professionals anticipate and avoid potential problems before construction begins, minimizing expensive delays and rework. Navisworks Manage combines spatial coordination with the project schedule to deliver 4D simulation and analysis. Entire project models can be published and freely viewed in NWD and DWF™ file formats.

What Is New in This Release?

Autodesk Navisworks Manage 2012 contains many new features and enhancements.

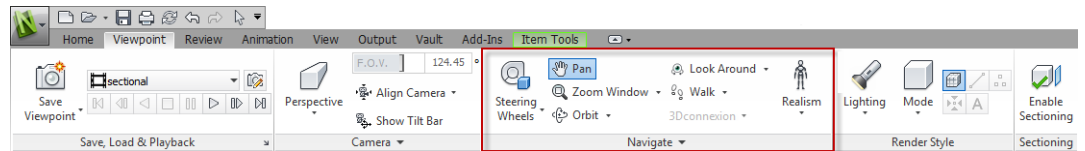
Installation

The installation screen provides links to the installation options, deployment options, installation tools and utilities. You also have the option of selecting the DWG file readers that require installation, plus the exporter plugins that you require and the Autodesk Navisworks Freedom viewer.

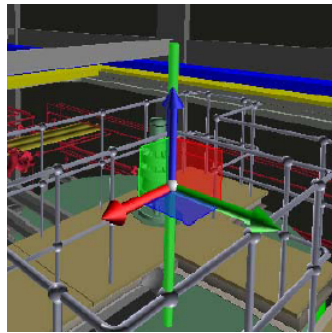
User Interface

Easy access to commonly used review and navigation tools to increase review productivity.

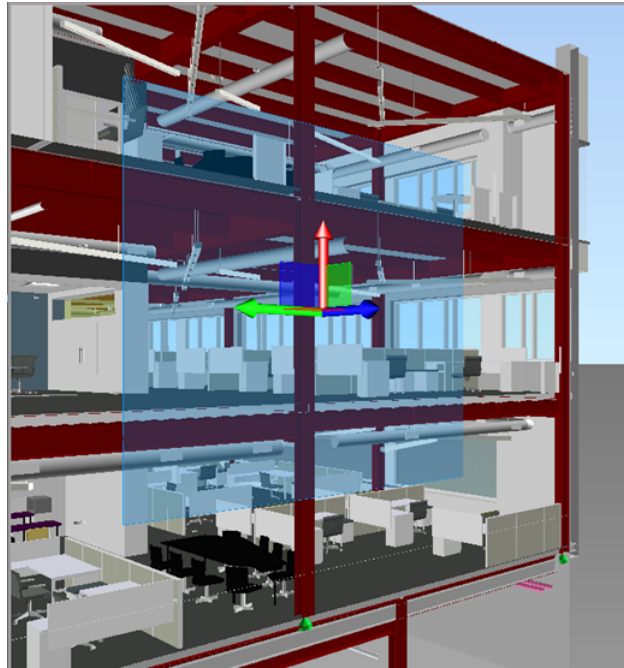
- The **Viewpoint** tab now includes the **Navigate** pane, providing access to tools such as walk, pan, zoom, and orbit; **SteeringWheels** tracking menus, **3Dconnexion** 3D mouse, and the realism settings.



- The gizmos have been updated, making it easier to manipulate objects and section planes.



- Section planes have also been enhanced to provide greater visual feedback of their position and orientation.

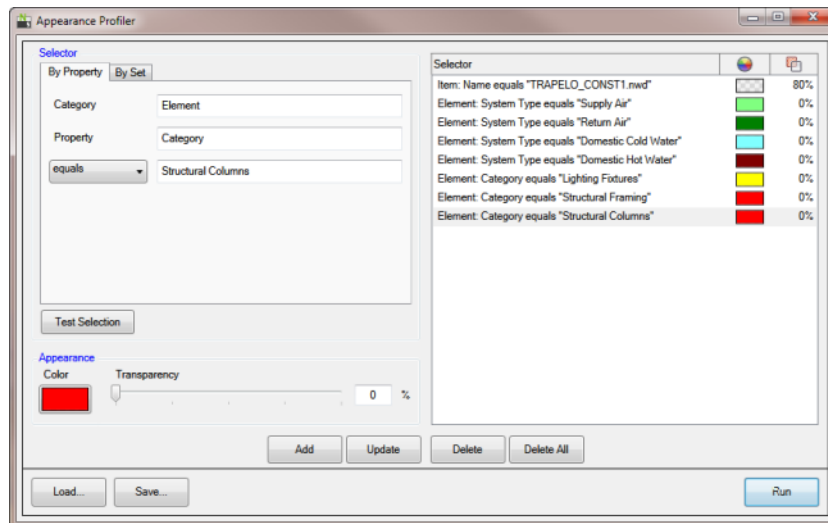


- Selection sets can now be created faster with the addition of the **Save Selection**  option (**Home** tab ► **Select & Search** panel).

Appearance Profiler

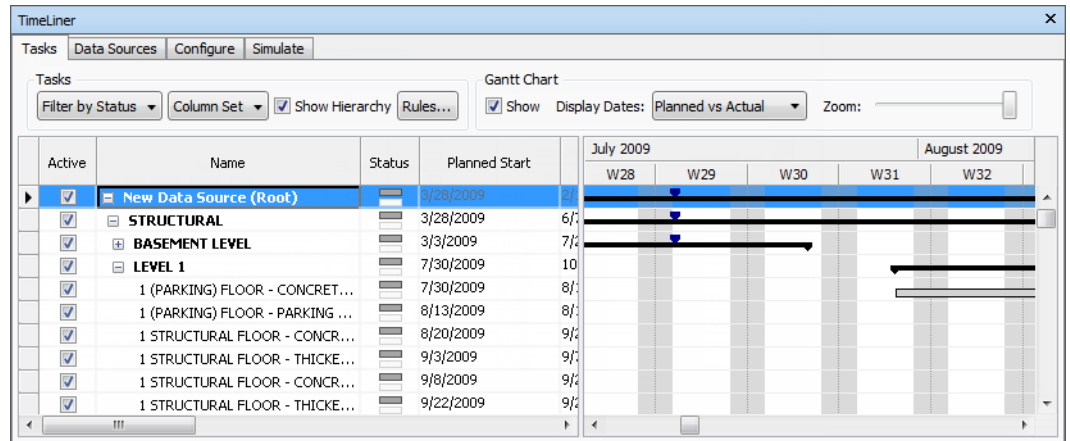
The **Appearance Profiler** allows you to set up custom appearance profiles based on sets (search and selection) and property values, and use them to color-code objects in the model to differentiate system types and visually identify their status. Appearance profiles can be saved and used on other projects, or shared between other Autodesk Navisworks users.

See [Appearance Profiler](#) (page 437).



TimeLiner

The **TimeLiner** tool has been enhanced to provide you with greater control and flexibility when working with 4D simulations. A new interface and integrated, editable Gantt chart allow you to create, edit and communicate your construction schedule more effectively. The new **TimeLiner** API allows you to extend and tailor the 4D toolset to meet your project and business requirements.



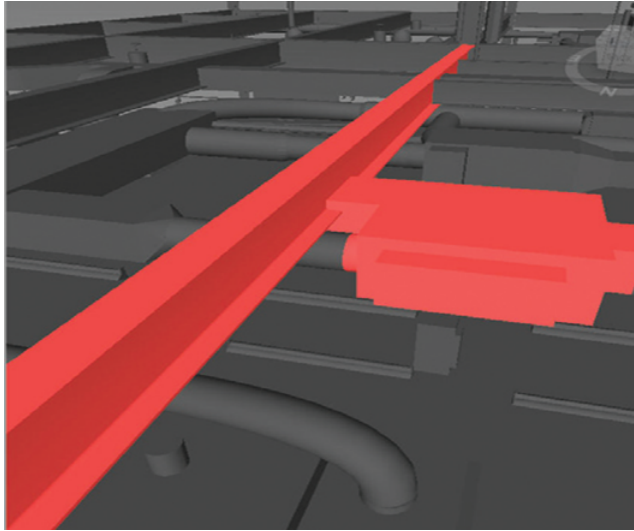
- The **Tasks**, **Gantt View** and **Rules** tab are now consolidated into a single **Tasks** tab.
- A column chooser allows you to select your preferred grid layout from three default options or by choosing your own columns.
- The interactive **Gantt Chart** lets you manipulate dates in the schedule by dragging and positioning tasks, start and end dates.
- A Gantt View is now available during 4D simulations on the **Simulate** tab.
- An ability to quickly add and edit tasks in a project schedule.
- An ability to filter tasks by their status.
- An ability to import / export **TimeLiner** rules for re-use.
- New **TimeLiner** .NET API. Using the API you can now obtain a list of tasks, task types, simulation types and data sources; add, edit and delete tasks, task types, simulation types and data sources and modify their properties; subscribe to events that will be triggered when the GUI or other API users make changes to tasks, task types, simulation types and data sources.

See [Overview of the TimeLiner Tool](#) (page 607).

Clash Detective

The **Clash Detective** tool has a number of improvements to offer better management of your clashes through to resolution. Workflow tools allow you to assign ownership of clashes and improved report tools provide you with a

batch summary of the status of your clashes and the ability to export a formatted report for direct use in Microsoft® Excel. Better visualization of clashes is supported by a new wireframe view and the **Focus on Clash** feature.



- New **Assigned To** clash attribute, enabling you to assign clashes and clash groups to an individual or trade, and keep track of the clashes through the **Results** list and **Clash Reports**.
- Summary information of the result statuses for each of the clash tests is now displayed on the **Batch** tab.
- Summary information for the current clash test is now displayed at the top of the **Clash Detective** tabs.
- Automatic notification of any model changes that may invalidate your clash results.
- New tabular HTML clash report that opens directly in Microsoft Excel.
- The option to view clash results using wireframe mode.
- The ability to reset a viewpoint to focus on the clash results again.

See [Overview of Clash Detective Tool](#) (page 669).

Autodesk File Format Support

Autodesk Navisworks now offers 2D DWF and multi-sheet DWF support, allowing you to open, review and explore your 2D datasets alongside your 3D

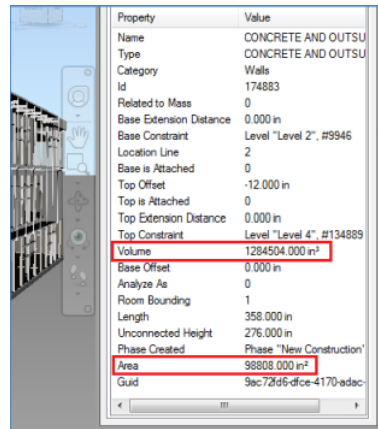
models. Importantly, the 2D view is integrated with the 3D environment - this enables you to select a component in the 3D model and then to find and review the same component in a 2D representation (such as a floor plan or section) providing you with the most appropriate view of the data for the task that you are undertaking. When working with the FBX visualization file format you can now achieve an accurate transfer of materials, textures and lights when importing or exporting data between Autodesk Navisworks and other FBX compatible applications.

- Support for opening 2D/3D DWF and DWFX files. See [DWF File Reader](#) (page 179).
- Support for exporting 3D DWF and DWFX files. See [Export 3D DWF/DWFX Files](#) (page 489).
- Multi-sheet file support. See [2D and Multi-Sheet Files](#) (page 213).
- 2D/3D Object Association support. See [Find All Sheets and Models Containing the Selected Object](#) (page 356).
- FBX consistent material support for lights, materials, and textures. See [FBX File Reader](#) (page 801).

Extended Support for Revit

A number of interoperability improvements to Revit / Autodesk Navisworks workflows increases your productivity when working with both applications.

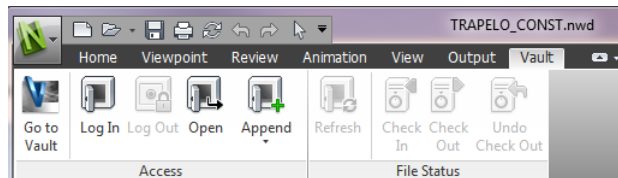
- Revit SwitchBack allows you to quickly transition between views in Autodesk Navisworks and Revit for easy navigation and location of elements. See [Revit SwitchBack](#) (page 436).
- Support for Revit Construction modelling enables you to pass your construction parts into Autodesk Navisworks for 4D simulation.
- Support for Revit linked files.
- Support for Revit split regions.
- Support for Revit properties including areas, volumes and points.



See [Revit File Exporter](#) (page 200).

Vault Integration

Autodesk Navisworks now offers integration with the Autodesk Vault data management toolset. Autodesk Vault offers a comprehensive environment for managing the large volume of data that is generated on your projects.



- Retrieve/save data.
- Check in/check-out data.
- Manage file versioning and the relationship between NWF files and design data.

See [Use the Autodesk Vault Add-In](#) (page 723).

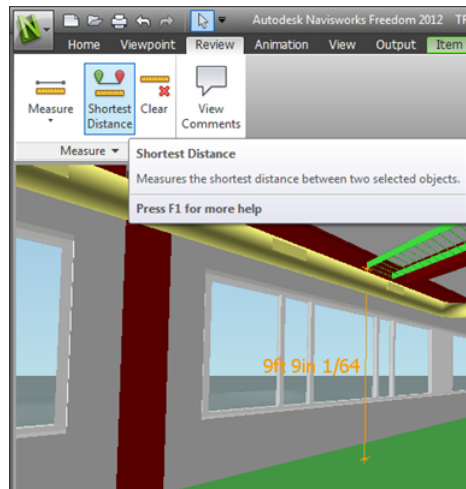
General Integration Enhancements

- Support for Google Sketchup v7 .skp files with backwards compatibility.
- **TimeLiner** now offers support for Primavera P6 v7 web services.
- Support for Pro/Engineer .prt, .asm, .g and .neu file formats.

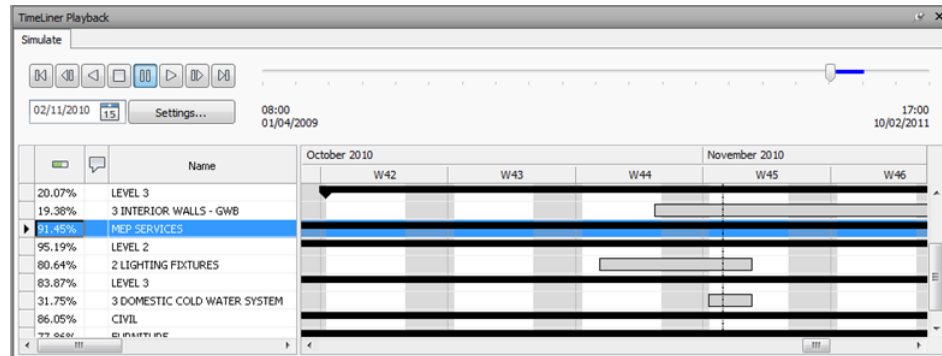
- Point cloud server support. Autodesk Navisworks now supports the retrieval of data from external point cloud engines for display within your Autodesk Navisworks model. The tool is implemented as an extension to the existing NWCreate API. A simple generic example and a customized example demonstrating connection to Z+F LFM server are available within the NWCreate API resources.

Autodesk Navisworks Freedom 2012 Enhancements

- The **Review** tab now contains **Measure** tools supporting field access to dimensioning and area calculation.



- You can now open 2D DWF, and multi-sheet DWF files, as well as NWD files.
- A Gantt View is now available during 4D **TimeLiner** simulations.



Miscellaneous Enhancements

- Enhanced support for the
- 3D mouse through an extended interface. See [3Dconnexion 3D Mouse](#) (page 298).
- Communication Centre now supports live updates.
- New avatars to be used in a variety of roles ranging from construction workers and safety professionals, to office workers. Since avatars can vary per viewpoint, you can easily show how project stakeholders will interact with a specific phase of the project in the relevant context.



- Ongoing implementation of Autodesk Navisworks .NET API.

How to Get Assistance

There are various ways to find information about how to use this program, and multiple resources are available.

Find Information Using InfoCenter

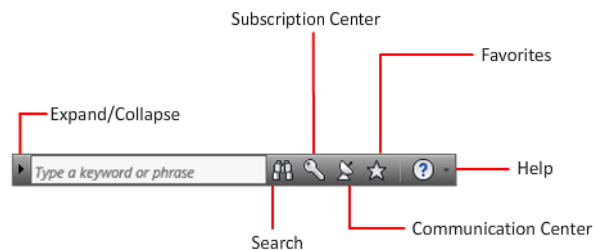
You can use InfoCenter to search Autodesk Navisworks help file for information. You can also easily access product updates and announcements.

Overview of InfoCenter

You can use InfoCenter to search for product-related help, display the Subscription Center panel for subscription services, display the Communication Center panel for product updates and announcements, and display the Favorites panel to access saved topics.

You can use InfoCenter to:

- Search for information in the main product Help through keywords (or by entering a phrase)
- Access subscription services through Subscription Center panel
- Access to product-related updates and announcements through Communication Center panel
- Access saved topics through Favorites panel



To display the InfoCenter box in a collapsed state, click the arrow to its left.



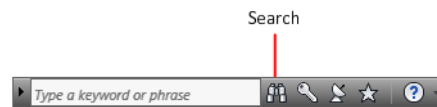
To rearrange the topics displayed on a panel

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.
 - In the InfoCenter box, click the Communication Center button.
 - In the InfoCenter box, click the Favorites button.
 - 2 Click and drag a category or group header to the desired position.
- TIP** To keep the Subscription Center, Communication Center, and the Favorites panel expanded, click the push pin icon in the bottom-right corner of the panel.

NOTE You can rearrange categories within a group, but you cannot move them into other groups.

Search for Information

You can enter keywords or a phrase in the **InfoCenter** box to search for information.



When you enter keywords or a phrase in the **InfoCenter** box, you search the contents of the main Autodesk Navisworks Help file.

Keyword searches produce better results. The results are listed on the Help Search tab. Click a topic to display it in help.

When you use InfoCenter to search for information, you can use the following special symbols in your query to refine or expand it. These symbols can be used alone or can be combined.

Symbol	Description
*	Replaces one or more characters when used at the beginning, middle, or end of a word. For example, “*lish”, “p*lish”, and “pub*” will find “publish”. Also, “anno*” will find “annotative”, “annotation”, “annoupdate”, “annoreset”, and so on.

Symbol	Description
?	Replaces a single character. For example, "cop?" will find "copy", but not "copybase".
~	Adds grammatical form variations to a keyword when added at the beginning or end of a word. For example, "plotting~" will find "plots", "plotted", and so on. Also, "~plot" will find "preplot", "replot", and so on.

When performing the exact phrase search, use double quotation marks (" ") to enclose words that must appear next to each other in the specified text string. For example, enter "**specify units of measurement**" to find only topics with all those words in that order. You can also use the previously mentioned symbols in a text string that is enclosed in double quotation marks.

To search the main Help file for information

- 1 In the InfoCenter box, enter a keyword or phrase.
- 2 Click the Search button.

The main Help file opens, and the search results are listed on the Help Search tab.

Access Subscription Center

Subscription Center displays links to information about subscription services such as product enhancements, personalized web support from Autodesk technical experts, and self-paced e-Learning.

If you are a subscription member, you can access subscription services by clicking the **Communication Center**  button in the **InfoCenter** box, and then clicking a **Subscription Center** link. To learn more about Autodesk subscription membership, visit <http://www.autodesk.com/subscriptioncenter>.

About Subscription Center

With Autodesk Subscription, you get the latest releases of Autodesk software, incremental product enhancements, personalized web support from Autodesk

technical experts, and self paced e-Learning. Subscription services are available to subscription members only.

By clicking the **Communication Center**  button in the **InfoCenter** box, members have access to the following options (under **Subscription Center**):

- **Subscription status.** Checks your subscription status.
- **Create support request.** Provides direct one-to-one communication with Autodesk support technicians. You receive fast, complete answers to your installation, configuration, and troubleshooting questions.
- **View support requests.** Tracks and manages your questions and responses through Autodesk's state-of-the-art support system.
- **Edit Subscription Center profile.** Sets up and maintains your subscription account.
- **View e-Learning catalog.** Features interactive lessons organized into product catalogs.
- **e-Learning Lessons.** (For subscription members only.) Each lesson is 15-30 minutes and features hands-on exercises, with an option to use a simulation instead of the software application. You can use an online evaluation tool that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.

Subscription Resources and Privacy

Subscription resources provide interactive product features over the Internet. Each time you access subscription resources (such as e-Learning or Create Support Request) from **Communication Center** in an Autodesk product, product information (such as the serial number, version, language, and the subscription contract ID) is sent to Autodesk for verification that your product is on subscription.

Autodesk compiles statistics using the information sent to subscription resources to monitor how they are being used and how they can be improved. Autodesk maintains the information provided by or collected from you in accordance with Autodesk's published privacy policy, which is available at <http://www.autodesk.com/privacy>.

To open the Subscription Center

- 1 Click the **Communication Center**  button in the **InfoCenter** box.
- 2 On the **Communication Center** panel, under **Subscription Center**, click the subscription resource you want to access.

NOTE Subscription Center is not available to all product users. If subscription resources are not available in your product, your product is not entitled to subscription benefits.

Manage Files with Autodesk Vault

If you are a subscription customer, you have access to Autodesk Vault, a file management tool that provides a repository where documents and files are stored and managed.

Autodesk Vault gives you more power to manage files and track changes. Versioned copies of master files are maintained, allowing you to easily revert to earlier versions of files. You can check files out for editing and later check them back in. The master copy is never directly edited.

Autodesk Vault consists of two required components: the Autodesk Data Management Server and the Vault Client. Optionally, you can also install the Vault Office Add-in.

For information about using the Vault, refer to the Vault Help system.

TIP The main components for the Autodesk Vault can be downloaded from the Autodesk Subscription site.

Use Communication Center

Communication Center provides up-to-date product information, software updates, product support announcements, and other product-related announcements.

Overview of Communication Center

You can click the Communication Center button to display links to information about product updates and announcements, and may include links to RSS feeds.

Whenever new information is available, **Communication Center** notifies you by displaying a balloon message below the **Communication Center** button in the **InfoCenter** box.

Communication Center provides the following types of announcements:

- **Autodesk Channels:** Receive support information, product updates, and other announcements (including articles and tips).
- **RSS Feeds.** Receive information from RSS feeds to which you subscribe. RSS feeds generally notify you when new content is posted. You are automatically subscribed to several default RSS feeds when you install the program.
- **Product Support Information.** Get breaking news from the Product Support team at Autodesk, including when Live Update maintenance patches are released.
- **Subscription Announcements.** Receive subscription announcements and subscription program news, as well as links to e-Learning Lessons, if you are an Autodesk subscription member (available in countries/regions where Autodesk subscriptions are offered).

For more information about Autodesk Subscription, see [Access Subscription Center](#) (page 13).

- **Articles and Tips.** Be notified when new articles and tips are available on Autodesk websites.
- **Live Update Maintenance Patches.** Receive automatic notifications whenever new maintenance patches are released from Autodesk.
- **Featured Technologies and Content.** Learn more about third-party developer applications and content.

You can customize the items that display on the **Communication Center** panel. For more information, see [Specify InfoCenter Settings](#) (page 23).

Communication Center Online Policy

Communication Center is an interactive feature that must be connected to the Internet in order to deliver content and information. Each time Communication Center is connected, it sends your information to Autodesk so that you receive the correct information. All information is sent anonymously to Autodesk to maintain your privacy.

Communication Center sends the following information to Autodesk:

- Product name (in which you are using Communication Center)
- Product release number
- Product language
- Country/region (specified in the Communication Center settings)

- Your unique Customer Involvement Program (CIP) ID if you are participating in the CIP program

Autodesk compiles statistics using the information sent from **Communication Center** to monitor how it is being used and how it can be improved. Autodesk maintains information provided by or collected from you in accordance with the company's published privacy policy, which is available at <http://www.autodesk.com/privacy>.

To open Communication Center

- In the **InfoCenter** box, click the **Communication Center**  button.

To receive new information notifications

- Click the link in the balloon message to open the article or announcement.

Save and Access Favorite Topics

You can click the Favorites button to display saved links to topics or web locations.



Any link that displays on the **Subscription Center** or **Communication Center** panel can be marked as a favorite.

A link marked as a favorite displays a star icon on the **Subscription Center** panel or the **Communication Center** panel.

To display the InfoCenter Favorites panel

- In the InfoCenter box, click the Favorites button.

NOTE The links displayed on the Favorites panel are organized into the same groups or categories from which they were added.

To save a link in InfoCenter as a favorite

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.

- In the InfoCenter box, click the Communication Center button.
- 2 Click the star icon that is displayed next to the link that you want to save as a favorite.

To remove a favorite link from the InfoCenter Favorites panel

- 1 In the InfoCenter box, click the Favorites button to display the Favorites panel.
- 2 Click the star icon that is displayed next to the link that you want to remove from the Favorites panel.

Use the Help System

You can click the Help button to display topics in Help.



You can get much more benefit from the Help system when you learn how to use it efficiently. You can quickly find general descriptions, procedures, details about dialog boxes and palettes, or definitions of terms.

The Help system contains complete information about using this program. In the **Help** window, you use the left pane to locate information. The tabs above the left pane give you several ways for finding the topics you want to view. The right pane displays the topics you select.

To display topics in Help

- In the InfoCenter box, click the Help button.

How Help Topics Are Organized

Most topics in this Help system have three tabs above the right pane of the Help window. The tabs display different types of information.

- **Concept tab.** Describes a feature or function. When you click the **Concept** tab, the Help Contents list in the left pane of the Help window expands and highlights the current topic. The **Contents** tab displays the

structure of the Help on that topic. You can easily display nearby topics by clicking them in the list.

- **Procedure tab.** Provides step-by-step instructions for common procedures related to the current topic. After displaying a procedure, you can click the **Procedure** tab to redisplay the current list of procedures.
- **Quick Reference tab.** Lists reference information related to the current topic.

When you click a different tab, the topic remains the same. Only the type of information displayed—concept, procedures, or quick reference links—is different.

Search in Help

Use the Help **Search** tab to find relevant topics based on keywords that you enter.

The basic search rules are as follows:

- Type your keywords in uppercase or lowercase characters; searches are not case-sensitive.
- Search for any combination of letters (a-z) and numbers (0-9).
- Do not use punctuation marks such as a period, colon, semicolon, comma, hyphen, and single quotation marks; they are ignored during a search.
- Group the elements of your search using double quotation marks or parentheses to set each element apart.

Use Wild Card Characters

You can use the following wild card characters in any keyword:

Symbol	Description
*	Replaces one or more characters when used at the beginning, middle, or end of a word. For example, <code>"*lish"</code> , <code>"p*lish"</code> , and <code>"pub*"</code> will all find <code>"publish"</code> . Also, <code>"anno*"</code> will find <code>"annotative"</code> , <code>"annotation"</code> , <code>"annoup-date"</code> , <code>"annoreset"</code> , and so on.

Symbol	Description
?	Replaces a single character. For example, "cop?" will find "copy", but not "copy-base".
~	Expands the tense of the word at the beginning or end of a word. For example, "plotting~" will find "plots", "plotted", and so on. Also, "~plot" will find "preplot", "replot", and so on.

Search for Phrases

When searching for a phrase, use double quotation marks (" ") to enclose words that must appear next to each other in the specified sequence. For example, enter "specifying units of measurement" to find only topics with all those words in that order. If you don't use the quotation marks around that text, Help finds all topics containing any one of the listed words, that is, all topics containing "specifying", all topics containing "units", all topics containing "of", and all topics containing "measurement".

TIP If you can't find the information you need through a search, try using the **Contents** tab.

Use Boolean Operators

With the AND, OR, NOT, and NEAR operators, you can precisely define your search by creating a relationship between search terms. The following table shows how you can use each of these operators. If no operator is specified, AND is used. For example, the query spacing border printing is equivalent to spacing AND border AND printing.

Search for	Example	Results
Both terms in the same topic	"tree view" AND "palette"	Topics containing both the words "tree view" and "palette"
Either term in a topic	viewpoint OR animation	Topics containing either the word "viewpoint" or the word "animation" or both

Search for	Example	Results
The first term without the second term	nwd NOT nwc	Topics containing the word "NWD", but not the word "NWC"
Both terms in the same topic, close together	user NEAR menu	Topics containing the word "user" within eight words of the word "menu"

NOTE The |, &, and ! characters do not work as Boolean operators. You must use AND (also +), OR, and NOT (also -).

Find Information in Help Topics

The tabs on the left side of the Help window provide different methods for finding information.

Contents Tab

- Presents an overview of the available documentation in a list of topics and subtopics.
- Allows you to browse by selecting and expanding topics.
- Provides a structure so you can always see where you are in Help and quickly jump to other topics.

Index Tab

- Displays an alphabetical list of keywords related to the topics listed on the **Contents** tab.
- Accesses information quickly when you already know the name of a feature, command, or operation, or when you know what action you want the program to perform.

Search Tab

- Provides a keyword search of all the topics listed on the **Contents** tab.
- Accepts the Boolean operators AND (+), OR, NOT (-), and NEAR.

- Accepts the wild cards *, ?, and ~.
- Allows you to perform a search for a phrase when the phrase is enclosed in double quotes.
- Displays a ranked list of topics that contain the word or words entered in the keyword field.
- Arranges the results alphabetically by title or by location if you click on the **Title** and **Location** column headings.

To find a specific word or phrase in the currently displayed Help topic

- 1 Click in the topic text and press CTRL+F.
- 2 In the **Find** text box, enter a keyword or phrase.
- 3 Click **Next**. If the keyword or phrase is located, the topic scrolls to display the result.

Print Help Topics

The quickest way to print the current topic is to right-click within the topic and click Print.

The **Print** button on the **Help** toolbar provides these print options:

- Print the selected topic (recommended)
- Print the selected heading and all subtopics

NOTE When you select the second option, you may get numerous printed pages, depending on how many subtopics the currently selected topic contains.

To print a Help topic

- 1 Display the topic you want to print.
- 2 Right-click in the topic pane. Click **Print**.
- 3 In the **Print** dialog box, click Print.

To print a selected heading and all subtopics

- 1 Display the topic you want to print and make sure that the **Contents** tab is displayed.
- 2 On the **Help** toolbar, click **Print**.

- 3 In the **Print Topics** dialog box, click **Print the Selected Heading and All Subtopics**.
- 4 Click **OK**.

Show and Hide the Contents Pane

You can control the size of the Help window.

Use the **Hide**  button on the **Help** toolbar to shrink the **Help** window to a compact size by hiding the pane that contains the **Contents**, **Index**, and **Search** tabs. The compact window size is best for displaying procedures while you work.

Use the **Show**  button to expand the **Help** window to display the pane that contains **Contents**, **Index**, and **Search** tabs. The expanded window size is best for locating and displaying conceptual and reference information.

Specify InfoCenter Settings

You can specify general and **Communication Center** settings in the **InfoCenter Settings** dialog box.

In the InfoCenter Settings dialog box, you can specify the following settings:

- **General.** Your current location, frequency for checking new online content and option to turn on or off animated transition effects for the InfoCenter panels.
- **Communication Center.** Set the maximum age of the articles displayed on the **Communication Center** panel.
- **Autodesk Channels.** Channels to display in the Communication Center panel as well as the number of articles to display for each channel.
- **Balloon Notification.** Notifications for new product information, software updates, and product support announcements. Also, you can customize the transparency and the display time of the balloon.
- **RSS Feeds.** RSS feed subscriptions. You can add or remove RSS feeds. RSS feeds generally notify you when new content is posted.

To specify the channels to display in the Communication Center panel

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.
 - In the InfoCenter box, click the Communication Center button.
 - In the InfoCenter box, click the Favorites button.
- 2 Click the **InfoCenter Settings**  button.
- 3 In the InfoCenter Settings dialog box, in the left pane, click Autodesk Channels.
- 4 In the right pane, select or clear the channels you want to display in the Communication Center panel.
- 5 Click OK.

To specify InfoCenter balloon notification settings

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.
 - In the InfoCenter box, click the Communication Center button.
 - In the InfoCenter box, click the Favorites button.
- 2 Click the **InfoCenter Settings**  button.
- 3 In the InfoCenter Settings dialog box, in the left pane, click Balloon Notification.
- 4 In the right pane, select or clear the options to turn balloon notification on or off.
- 5 Enter the number of seconds to set the length of time for balloon notifications to display.
- 6 Enter the transparency value of the balloon or set the value using the slider.
- 7 Click OK.

To add an RSS feed to Communication Center

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.
 - In the InfoCenter box, click the Communication Center button.
- 2 Click the **InfoCenter Settings**  button.
- 3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.

- 4 In the right pane, do one of the following:
 - Click Add.
 - Right-click anywhere in the right pane. Click Add.
- 5 In the Add RSS Feed dialog box, enter the location of the RSS feed you want to add. Click Add.
- 6 In the InfoCenter - RSS Feed Confirmation dialog box, click Close.
- 7 Click OK.

To remove an RSS feed from Communication Center

- 1 Display a panel by doing one of the following:
 - In the InfoCenter box, click the Subscription Center button.
 - In the InfoCenter box, click the Communication Center button.
 - In the InfoCenter box, click the Favorites button.
- 2 Click the **InfoCenter Settings**  button.
- 3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.
- 4 In the right pane, do one of the following:
 - Click Remove.
 - Right-click an RSS feed. Click Remove.
- 5 In the InfoCenter - Remove RSS Feed dialog box, click Yes.
- 6 Click OK.

Get More Help

You can access several additional sources of help.

- **Use Communication Center.** Display the **Communication Center** panel for product updates and announcements.
- **Press F1.** Displays context-sensitive reference information.
- **Click the Help button in many dialog boxes.** Displays reference information for the dialog box.
- **View the product Readme.** Displays late-breaking information about this product.

Other resources help you get information about Autodesk products and assistance with your questions about this program.

- **Autodesk website.** Access <http://www.autodesk.com>.
- **Local support.** Check with your dealer or Autodesk country/region office.

Learn the Product

Training programs and products from Autodesk help you learn the key technical features and improve your productivity.

For the latest information about Autodesk training, visit <http://www.autodesk.com/training> or contact your local Autodesk office.

Autodesk Authorized Training Centers

The Autodesk® Authorized Training Center (ATC®) network delivers Autodesk-authorized, instructor-led training to design professionals who use Autodesk software. Autodesk Authorized Training Centers use experienced and knowledgeable instructors. More than 1,200 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.

To find a training center near you, contact your local Autodesk office or visit <http://www.autodesk.com/atc>.

Autodesk Official Training Courseware

Autodesk Official Training Courseware (AOTC) is technical training material developed by Autodesk. Designed for traditional 1/2-day to 5-day, instructor-led classroom training and used by Authorized Training Centers and other Autodesk partners, AOTC is well-suited for self-paced, stand-alone learning. The manuals cover key concepts and software functionality with hands-on, step-by-step, real-world exercises. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store at <http://www.autodesk.com/aotc>.

e-Learning

Autodesk e-Learning for Autodesk Subscription customers features interactive lessons organized into product catalogs. Each lesson is 20-40 minutes in length and features hands-on exercises, with an option to use a simulation of the product or the actual application. You can also use an online evaluation tool

that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.

If you are a member of Autodesk subscription, you can access e-Learning and other subscription services from within your Autodesk product.

For more information about how to access e-Learning in the product, see [Access Subscription Center](#) (page 13).

For more information about Autodesk subscription resources, visit <http://www.autodesk.com/subscriptioncenter>.

Autodesk Developer Network

The Autodesk Developer (ADN) program for ADN members provides support for full-time, professional developers who want to build software based on Autodesk products. As an ADN member, you will receive the business, software, support, and training you need to be successful. If you are a developer, visit <http://www.autodesk.com/adn>.

Autodesk Consulting

Autodesk Consulting provides services that help set up processes and provide critical training that will help increase productivity so you can capitalize on the power of your products. For more information on general consulting, systems integration, or custom training services, visit <http://www.autodesk.com/consulting>.

Partner Products and Services

Autodesk works together with thousands of software partners around the world. These partners provide products and services that enhance Autodesk products for design professionals. Visit the Partner Products & Services page at <http://www.autodesk.com/partnerproducts> for a list of resources available for your Autodesk product and your industry.

View the Product Readme

You can find late-breaking information about this software in the Readme.

It is suggested that you read through the Autodesk Navisworks Readme for information about recommended hardware, updated installation instructions, and known software problems. The Readme file is available from the product's program group on the Windows Start menu.

Join the Customer Involvement Program

You are invited to help guide the direction of Autodesk design software.

If you participate in the Customer Involvement Program (CIP), specific information about how you use Autodesk Navisworks is forwarded to Autodesk. This information includes what features you use the most, problems that you encounter, and other information helpful to the future direction of the product.

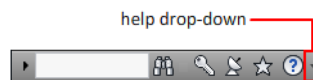
See the following links for more information.

- Learn more about the Autodesk Customer Involvement Program: <http://www.autodesk.com/cip>
- Read the Autodesk Privacy Statement: <http://www.autodesk.com/cipprivacy>

When you join, you will be able to view reports that can help you optimize your use of Autodesk Navisworks.

To turn the CIP on or off

- 1 On the InfoCenter toolbar, to the right of the Help button, click the drop-down arrow.



- 2 Click Customer Involvement Program.
- 3 In the Customer Involvement Program dialog box, select to start or stop participating.
- 4 Click OK.

Installation

2

Quick Start to Stand-Alone Installation

This section provides step-by-step instructions about how to prepare, and then install Autodesk Navisworks.

Stand-alone installation is recommended for individual users or small groups. The key point is that you will repeat the installation process on each computer. For a stand-alone license this is the only valid installation type, but it can also be used with a multi-seat stand-alone or network license.

If you have never installed the product before, you should familiarize yourself with the entire installation process and options before beginning.

For information about installing network-licensed or multi-seat stand-alone versions of the program, see [Install Autodesk Navisworks for Multiple Users](#) (page 44).

Prepare for Installation

To prepare for installation, you should review the system requirements, understand administrative permission requirements, locate your Autodesk Navisworks Manage 2012 serial number and product key, and close all running applications.

Complete these tasks, and you are ready to begin installing Autodesk Navisworks Manage 2012.

NOTE It is also recommended that you install Microsoft .Net Framework 4.0 before installing the product. See [Install Microsoft .Net Framework 4.0](#) (page 31).

System Requirements for Stand-Alone Installation

The first task you need to complete is to make sure that your computer meets the minimum system requirements. If your system does not meet these requirements, problems can occur, both within Autodesk Navisworks and at the operating system level.

Whether your Windows operating system is the 32-bit or the 64-bit version, the version is automatically detected during installation.

See the following table for hardware and software requirements.

Hardware and software requirements for client machine	
Hardware/Software	Requirement
Operating System	Microsoft® Windows 7 (32-bit or 64-bit) Home Basic, Home Premium, Professional, Enterprise, or Ultimate (recommended)
	Microsoft® Windows Vista® SP2 (32-bit or 64-bit) Home Premium, Business, Enterprise, or Ultimate
	Microsoft® Windows XP SP3 (32-bit) Home, or Professional
	Microsoft® Windows XP SP2 (64-bit) Professional
Web browser	Microsoft® Internet Explorer® 7.0 or later
Processor	AMD Athlon™, 3.0 GHz or faster (minimum); Intel® Pentium® 4, 3.0 GHz or faster (recommended) - with SSE2 technology
Memory (RAM)	512 MB (minimum); 2 GB or more (recommended)
VGA Display	1024 x 768 with true color (minimum)
	1280 x 1024 32-bit color video display adapter with true color (recommended)

Hardware and software requirements for client machine	
Graphics Card	Direct3D 9® and OpenGL® capable graphics card with Shader Model 2 (minimum)
Hard disk	11 GB free disk space for installation
Pointing device	Microsoft® Mouse-compliant pointing device
DVD-ROM	Any speed (for installation only)
Optional hardware	Printer or plotter
	Modem or access to an Internet connection
	Network interface card

Install Microsoft .Net Framework 4.0

Autodesk Navisworks Manage 2012 requires Microsoft .Net 4.0 to be installed prior to product installation.

Most Windows updates should include Microsoft .NET Framework 4, however, for old versions of Windows, you can download and install Microsoft .NET Framework 4 redistributables from: <http://www.microsoft.com/downloads/en/details.aspx?FamilyID=0a391abd-25c1-4fc0-919f-b21f31ab88b7> or install Microsoft .Net Framework 4 from the following directory in the installation media:
 \3rdParty\NET\4\wcu\dotNetFramework\.

The **Installation** wizard will install Microsoft .Net Framework 4 if it detects that required updates have not been installed.

NOTE If the **Installation** wizard prompts you to install the Microsoft .NET 4.0 Framework, the .NET 4.0 Framework installer may prompt you to first install the Windows Imaging Component (WIC). This can occur if you do not have the latest Microsoft Windows updates or service packs. You are most likely to need WIC if you are running Windows XP SP2 without certain Windows Updates installed. If required, the Microsoft WIC installers are available at the following locations:

- 32-bit installer: <http://www.microsoft.com/downloads/en/details.aspx?FamilyID=8e011506-6307-445b-b950-215def45ddd8&displaylang=en>
 - 64-bit installer: <http://www.microsoft.com/downloads/en/details.aspx?FamilyID=f64654ac-6e26-41d9-a90a-0e7783b864ee>
-

Understand Administrative Permission Requirements

To install Autodesk Navisworks, you must have administrator permissions.

You do not need to have domain administrative permissions. See your system administrator for information about administrative permissions.

To run Autodesk Navisworks, you do not need administrator permissions. You can run the program as a limited user.

Locate Your Autodesk Navisworks Serial Number and Product Key

When you install Autodesk Navisworks, you are prompted for your serial number and product key in the **User and Product Information** page.

The serial number must contain a three-digit prefix followed by an eight-digit number. The product key consists of five digits.

The serial number and product key are located on the outside of the product packaging, or in the email you received if you downloaded your product. Make sure to have these available before you install the program so that you don't have to stop in the middle of the installation. Also make sure to have these available before you activate the program.

The information you enter is permanently retained with the product. Because you can't change this information later without uninstalling, take care when

entering the information. To review this product information later, in the **InfoCenter** box, click the down arrow next to the **Help** button ► **About** Autodesk Navisworks Manage 2012.

NOTE If you have lost your serial number or product key, contact your local Autodesk office for assistance.

Avoid Data Loss During Installation

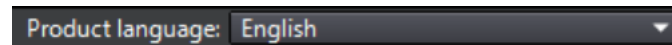
The Autodesk Navisworks installation process may stop if some applications (such as Microsoft® Outlook® or virus-checking programs) are running.

Close all running applications to avoid possible data loss.

Choose a Language

You can select a different language for installation instructions, and a language for individual product installations in the same install process.

When you start the installation process, the installer automatically determines your operating system language. If a supported language is detected, your install pages are displayed in that language. If you want to change that language, you select a different one from the Installer Language list on the first page of the Installation wizard.



NOTE Some products may not have multi-language support at the time of product release. Additional language support may be available later. Check <http://support.autodesk.com> for the availability of additional language packs.

Using Language Packs

Language packs support use of different languages in each Autodesk Navisworks product, including exporters. Pack names start with **NAVFREE_**, **NAVSIM_**, **NAVMAN_**, and **exporters_** respectively.

NOTE You must install at least one language pack for each product.

It is possible to install additional language packs to Autodesk Navisworks products later. You can manually install the required language packs by double-clicking on the language pack MSI file.

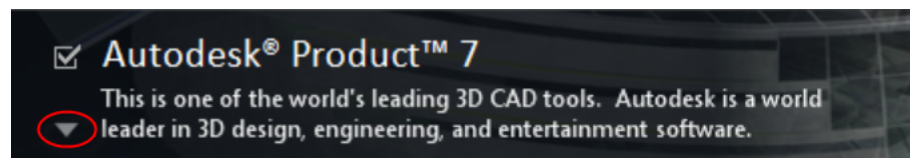
Language packs are located on the installation DVD and unpacked downloaded media under the **x86** folder for 32-bit products and under the **x64** folder for 64-bit products.

- Language packs for specific products are included in the **NAVFREE**, **NAVSIM**, **NAVMAN**, and **NWEXPORT** subfolders of **x86** and **x64** folders.
- Language packs for specific languages are included in the **en-US** (English), **de-DE** (German), **es-ES** (Spanish), **fr-FR** (French), **it-IT** (Italian), **ja-JP** (Japanese), **ko-KR** (Korean), **pt-BR** (Brazilian Portuguese), **ru-RU** (Russian) and **zh-CN** (Chinese PRC) subfolders of the product folders.

So, for example, to install the 32-bit French language pack for **Autodesk Navisworks Manage**, double-click `x86/NAVMAN/fr-FR/NAVMAN_LP.msi`.

Configuration

During the installation process, you choose either a typical installation (install the product with default settings), or a customized installation. On the Install ► **Configure Installation** page, for any product you have selected to install, you will see a triangular control for access to a configuration panel.



Click anywhere in the product box to open the configuration panel:

- **Installation Type.** If you choose a **Typical** installation (default settings) the product installs the most common application features. A **Custom** installation installs only the application features that you select from the **Select Features To Install** list. The available features will depend upon the product you are installing:

Redistributable ActiveX Control

Contains Autodesk Navisworks Redistributable ActiveX control.

API	Contains the Component Object Model interface for customizing and extending the Autodesk Navisworks functionality.
Example NWD files	Contains various feature sample files.
Batch Utility	Contains an add-in to run and schedule commands.
Program	Contains full set of Autodesk Navisworks files.
Sample RPCs	Contains several Rich Photorealistic Content files.

NOTE For Autodesk Navisworks exporter plugins, you can change which plugins are installed.

 Indicates the third-party software already installed on your computer.

 Indicates plugins for the third-party software, that are either not installed or have not been detected by the **Installation** wizard.

Select the check boxes next to all required plugins. If the third-party software has not been detected by the wizard, you can manually enable the plugins for it. Selecting the check box automatically opens the dialog box, which enables you to browse for the correct software installation directory.

- **Create the Desktop Shortcut.** Select the check box to create the desktop shortcut for Autodesk Navisworks.
- **Project and Site Folders.** Use the **Browse** buttons to select the directories that will contain Autodesk Navisworks settings that can be shared across an entire project site, or across a specific project group (see [Select the Project and Site Folders](#) (page 64)).
- **Service Packs.** If a service pack is available for your installation, you can include it in the installation.

After you have configured the settings as required, click the product name to close the configuration panel.

Install Multiple or Bundled Products

Some Autodesk packages are comprised of multiple products or are part of multi-product bundles.

The Installation wizard for packages that are comprised of multiple products gives you the option to choose which products you want to install.

In the Installation wizard, for packages containing multiple products, you can choose which products and languages you want to install. During the install process, you are informed whether a copy of the software is already installed. You are also warned if your system does not meet the minimum system requirements for the product. Each product name is displayed on its own tabbed panel; you can configure them individually.

If you purchased a package that is a multi-product bundle, such as an educational or institutional package, you may have a package that includes several Autodesk products. For these bundled packages, an Installer disc contains information for all the products in the package. The Installer disc helps you install all of the products.

Install and Run Autodesk Navisworks Manage 2012

To use the product, you must install the product, register and activate it, and then launch it.

You must have administrative permissions to install Autodesk Navisworks.

This section provides instructions for installing and activating Autodesk Navisworks for an individual user on a stand-alone computer.

For information about installing network-licensed or multi-seat stand-alone versions of the program, see [Install Autodesk Navisworks for Multiple Users](#) (page 44).

When you have successfully installed Autodesk Navisworks Manage 2012, you are ready to register your product and start using the program. To register the product, start Autodesk Navisworks Manage 2012 and follow the on-screen instructions. For more information, see [How do I register and activate Autodesk Navisworks?](#) (page 75)

NOTE Autodesk does not recommend or support the distribution of an Autodesk product using imaging software.

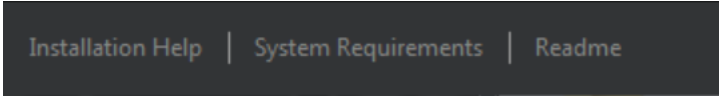
Install Autodesk Navisworks

The Autodesk Navisworks **Installation** wizard contains all installation-related material in one place.

From the **Installation** wizard, you can access user documentation, change the installer language, select a language-specific product, install supplemental tools, view support solutions, and learn about deploying your product on a network.

NOTE The 32-bit and 64-bit versions of Autodesk Navisworks Manage 2012 are on separate DVDs. Insert the appropriate Autodesk Navisworks DVD in the DVD drive to start the installation process. Follow the prompts to complete the installation. As long as the DVD is in the drive, you can access user documentation by clicking the **Installation Help** link.

- **Review installation documentation before you install.** It is recommended that you take the time to familiarize yourself with the complete installation process before you install Autodesk Navisworks. Documentation is accessible from links on the lower left corner of the installer.



Installation Help | System Requirements | Readme

- **Install Autodesk Navisworks Manage 2012.** From the **Installation** wizard, click **Install**. Follow the on-screen instructions to complete the installation.

Install Autodesk Navisworks Using Default Values

This is the fastest means of installing Autodesk Navisworks on your system.

Only default values are used which means it is a typical installation being installed to C:\Program Files\Autodesk\Navisworks Manage 2012.

To install Autodesk Navisworks using default values on a stand-alone computer

- 1 Close all running applications on your computer and start the **Installation** wizard.

- 2 On the **Installation** wizard, if required, select an alternate language for the **Installation** wizard from the **Installation Instructions** drop-down, and then click **Install**.
- 3 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed with the installation. Choose your country or region, click **I Accept**, and then click **Next**.

NOTE If you do not agree to the terms of the license and want to terminate the installation, click **Cancel**.

- 4 On the **Product Information** page, select **Stand-Alone** and enter your serial number and product key, then click **Next**.
- 5 On the **Configure Installation** page, select the products to install, and if required add a language pack(s) from the **Product Language** drop-down (see [Choose a Language](#) (page 33)).
- 6 If required, use the **Installation PathBrowse** button to select the drive and location where product will be installed.
- 7 Click **Install**. The wizard installs the products you selected using a **Typical** installation, which installs the most common application features. To see which features are included in a **Typical** installation, refer to [Typically Installed Features](#) (page 72).

NOTE By default, the **Installation** wizard automatically enables the exporter plugins for all third-party products already installed on your computer.

- 8 Click **Finish**.

Install Autodesk Navisworks Using Configured Values

With this installation method, you can fine-tune exactly what gets installed.

You can alter the license type, the installation type, the install path, and specify the location of the Project and Site folders.

To install Autodesk Navisworks using configured values on a stand-alone computer

- 1 Close all running applications on your computer and start the **Installation** wizard.

- 2 On the **Installation** wizard, if required, select an alternate language for the **Installation** wizard from the **Installation Instructions** drop-down, and then click **Install**.
- 3 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed with the installation. Choose your country or region, click **I Accept**, and then click **Next**.

NOTE If you do not agree to the terms of the license and want to terminate the installation, click **Cancel**.
- 4 On the **Product Information** page, select the **License Type** (**Stand-Alone** or **Network**) and enter your serial number and product key, then click **Next**.
- 5 On the **Configure Installation** page, select the products to install, and if required add a language pack(s) from the **Product Language** drop-down (see [Choose a Language](#) (page 33)).
- 6 Click the product name to open the configuration panel where you can review and change settings. See [Configuration](#) (page 34). After you have configured the settings as required, click the product name to close the configuration panel.
- 7 If required, use the **Installation PathBrowse** button to select the drive and location where product will be installed.
- 8 Click **Install**. The wizard installs the products you selected using your **Custom** installation settings.
- 9 Click **Finish**.

Launch Autodesk Navisworks

Assuming that you've followed all of the previous steps outlined in this Quick Start section, you can launch Autodesk Navisworks and start taking advantage of its new and updated features.

You can start Autodesk Navisworks in the following ways:

- **Desktop shortcut icon.** When you install Autodesk Navisworks, a shortcut icon is placed on your desktop. Double-click the Autodesk Navisworks icon to start the program.
- **Start button.** Click **Start** ► **All Programs** ► **Autodesk** ► Navisworks Manage 2012 ► Autodesk Navisworks Manage 2012.
- **Location where Autodesk Navisworks is installed.** If you have administrative permissions, you can run Autodesk Navisworks in the

location where you installed it. If you are a limited-rights user, you must run Autodesk Navisworks from the **Start** button or from the desktop shortcut icon. If you want to create a custom shortcut, make sure that the **Start In** directory for the shortcut points to a directory where you have write permissions.

NOTE When the product is started, by default, it uses the language that best matches the settings on your computer. You can also launch Autodesk Navisworks in another of the supported languages.

How to Launch Autodesk Navisworks in Another Language

To run Autodesk Navisworks in another of the installed languages, you need to add one of the language selector arguments to the desktop shortcut.

To run Autodesk Navisworks in another language

- 1 Right-click the Autodesk Navisworks desktop shortcut, and click **Properties** on the shortcut menu to open the Autodesk Navisworks **Properties** dialog box.
- 2 On the **Shortcut** tab, enter a space in the **Target** field after `..\roamer.exe`, and then enter one of the following arguments:
 - **-lang en-US** for English localization
 - **-lang de-DE** for German localization
 - **-lang es-ES** for Spanish localization
 - **-lang fr-FR** for French localization
 - **-lang it-IT** for Italian localization
 - **-lang ja-JP** for Japanese localization
 - **-lang ko-KR** for Korean localization
 - **-lang pt-BR** for Brazilian Portuguese localization
 - **-lang ru-RU** for Russian localization
 - **-lang zh-CN** for Chinese (PRC) localization
- 3 Click **OK** to save the changes.

Add or Remove Features

You can add or remove Autodesk Navisworks Manage 2012 features at any time. For example, you may have chosen a **Custom** installation option when you first installed Autodesk Navisworks, but now you want to add features that you did not install originally. Or you may no longer need to use all of the features that were installed originally.

You can add or remove features by using the Control Panel.

To add or remove features

- 1 Do one of the following:
 - (Windows XP) Click **Start > Settings > Control Panel > Add or Remove Programs**.
 - (Windows Vista and Windows 7) Click **Start > Control Panel > Programs and Features**.
- 2 From the list of programs, click Autodesk Navisworks Manage 2012, and then click **Change/Remove** (Windows XP) or **Uninstall/Change** (Windows Vista and Windows 7).

The Autodesk Navisworks Manage 2012 **Installation** wizard re-opens in **Maintenance Mode**.

- 3 Click **Add or Remove Features**. On the **Add/Remove Features** page, select a feature to install or uninstall. The icons to the left of the selections give you an indication of the action that will be taken.



Indicates a feature that was marked for installation will be in a typical installation.



Indicates a feature that is not currently scheduled for installation.



Indicates a feature that was not originally marked for installation, but was added to the installed feature list.



Indicates an installed feature that is chosen for removal.

NOTE If you need to revert to the Autodesk Navisworks Manage 2012 features that you selected in your original installation, click **Cancel**.

Click **Update**.

- 4 On the **Update Complete** page, you are informed when the updates have been performed. Click **Finish**.

Repair Autodesk Navisworks Manage 2012

If you accidentally delete or alter files that are required by Autodesk Navisworks Manage 2012, Autodesk Navisworks might not perform correctly, and you might receive error messages when you try to execute a command or find a file. You can attempt to fix this problem by repairing Autodesk Navisworks Manage 2012.

Repairing uses the features that were part of the installation type you chose when you initially installed the program.

To repair Autodesk Navisworks Manage 2012

- 1 Do one of the following:
 - (Windows XP) Click **Start > Settings > Control Panel > Add or Remove Programs**.
 - (Windows Vista and Windows 7) Click **Start > Control Panel > Programs and Features**.
- 2 From the list of programs, click Autodesk Navisworks Manage 2012, and then click **Change/Remove** (Windows XP) or **Uninstall/Change** (Windows Vista and Windows 7).

The Autodesk Navisworks Manage 2012 **Installation** wizard re-opens in **Maintenance Mode**.
- 3 Click **Repair or Reinstall**.
- 4 On the **Repair or Reinstall** page, click **Repair** Autodesk Navisworks Manage 2012. This option replaces all registry entries that Autodesk Navisworks initially installed and restores Autodesk Navisworks Manage 2012 to its default state. Click **Repair**.

NOTE Reinstall Autodesk Navisworks Manage 2012 repairs the registry and reinstalls all files from the original installation. Use this option if the **Repair** Autodesk Navisworks Manage 2012 option does not solve the problem.

- 5 On the **Repair Complete** page, click **Finish**.

Uninstall Autodesk Navisworks Manage 2012

When you uninstall Autodesk Navisworks Manage 2012, all components are removed. This means that even if you've previously added or removed

components, or if you've reinstalled or repaired Autodesk Navisworks Manage 2012, the uninstall removes all Autodesk Navisworks installation files from your system.

IMPORTANT Do not use registry cleaning programs or attempt to modify the registry entries yourself to uninstall Autodesk Navisworks Manage 2012. Failure to follow the official uninstall procedure will result in the inability to re-install the software .

To uninstall Autodesk Navisworks Manage 2012

- 1 Do one of the following:
 - (Windows XP) Click **Start > Settings > Control Panel > Add or Remove Programs**.
 - (Windows Vista and Windows 7) Click **Start > Control Panel > Programs and Features**.
- 2 From the list of programs, click Autodesk Navisworks Manage 2012, and then click **Change/Remove** (Windows XP) or **Uninstall/Change** (Windows Vista and Windows 7).

The Autodesk Navisworks Manage 2012 **Installation** wizard re-opens in **Maintenance Mode**.
- 3 Click **Uninstall**.
- 4 When informed that the product has been successfully uninstalled, click **Finish**.

NOTE Even though Autodesk Navisworks Manage 2012 is removed from your system, the software license remains. If you reinstall Autodesk Navisworks Manage 2012 at some future time, you will not have to register and re-activate the program.

Move to Autodesk Navisworks from a Previous Release

If you have a previous version of Autodesk Navisworks installed on your system, you can install Autodesk Navisworks Manage 2012 and keep other versions of the program on the same system. This is called a *side-by-side* installation.

If you've purchased an *upgrade* version of Autodesk Navisworks Manage 2012, you are required to uninstall the previous version within 120 days of installing

Autodesk Navisworks Manage 2012. See your license agreement for more information.

Install Autodesk Navisworks for Multiple Users

This section provides step-by-step instructions for installing network-licensed or multi-seat stand-alone versions of the Autodesk Navisworks Manage 2012.

Quick Start to Network Administration and Deployment

Network deployment of this program requires careful planning and execution.

Deployment installation is recommended for network administrators, using either a multi-seat stand-alone or network license. In the deployment process, the installation is configured once, stored on the network, and then distributed efficiently to users' computers.

If you are not familiar with network administration and deployment, you should familiarize yourself with the following topics before you attempt to deploy and administer the program over a network.

Deployment Preparation

To prepare for a deployment, you should take the time to review the following requirements and options.

System Requirements for a Deployment

This section contains the system requirements for the location of the administrative image that you create, the network license server, and the client workstation.

Before you begin installing the program on a network, make sure that your servers and client workstations meet the minimum recommended hardware and software requirements for a deployment.

Whether the Windows operating system is the 32-bit or the 64-bit version is automatically detected when installing Autodesk Navisworks. You can choose which versions of Autodesk Navisworks will be installed.

See the following tables for administrative image, license server, and client workstation system requirements.

Hardware and software requirements for the location of the administrative image

Hardware/Software	Requirement
Hard disk	6 GB

Hardware and software requirements for client machine

Hardware/Software	Requirement
Operating System	Microsoft® Windows 7 (32-bit or 64-bit) Home Basic, Home Premium, Professional, Enterprise, or Ultimate (recommended)
	Microsoft® Windows Vista® SP2 (32-bit or 64-bit) Home Premium, Business, Enterprise, or Ultimate
	Microsoft® Windows XP SP3 (32-bit) Home, or Professional
	Microsoft® Windows XP SP2 (64-bit) Professional
Web browser	Microsoft® Internet Explorer® 7.0 or later
Processor	AMD Athlon™, 3.0 GHz or faster (minimum); Intel® Pentium® 4, 3.0 GHz or faster (recommended) - with SSE2 technology
Memory (RAM)	512 MB (minimum); 2 GB or more (recommended)
VGA Display	1024 x 768 VGA with true color (minimum)

Hardware and software requirements for client machine	
	1280 x 1024 32-bit color video display adapter with true color (recommended)
Graphics Card	Direct3D 9® and OpenGL® capable graphics card with Shader Model 2 (minimum)
Hard disk	11 GB free disk space for installation
Pointing device	Microsoft® Mouse-compliant pointing device
DVD-ROM	Any speed (for installation only)
Optional hardware	Printer or plotter
	Modem or access to an Internet connection
	Network interface card

Hardware and software requirements for the network license server (Windows)

Hardware/Software	Requirement
Operating System	Microsoft® Windows 7 (32-bit or 64-bit) ■ Windows 7 Enterprise ■ Windows 7 Ultimate ■ Windows 7 Professional ■ Windows 7 Home Premium
	Microsoft® Windows Vista® SP2 or later (32-bit or 64-bit) ■ Windows Vista Enterprise ■ Windows Vista Ultimate ■ Windows Vista Business ■ Windows Vista Home Premium and Basic (32-bit)

Hardware and software requirements for the network license server (Windows)

Hardware/Software	Requirement
	■ Windows Vista Home Premium (64-bit)
	Microsoft® Windows XP SP2 or later (32-bit or 64-bit) ■ Windows XP Professional ■ Windows XP Home
	Windows Server SP1 or later (32-bit or 64-bit) ■ Windows 2008 Server R2 ■ Windows 2008 Server ■ Windows 2003 Server R2 ■ Windows 2003 Server
Computer/processor	Intel® Pentium® III or higher 450 Mhz (minimum)
Network interface card	Compatible with existing Ethernet network infrastructure NOTE The Network License Manager supports multiple network interface cards, but at least one must be an Ethernet card.
Communication protocol	TCP/IP NOTE The Network License Manager uses TCP packet types.
Browsers	Internet Explorer® 6.1 Internet Explorer 7 Internet Explorer 8
FLEXnet	11.9.0.0

Determine the Installation Type

When you set up your deployment, you specify the installation type by target platform and license type.

Target Platform

Specify either a 32-bit or 64-bit platform, depending on the operating system of the computers that will use the deployment. For some Autodesk products, you can install a 32-bit version on a 64-bit operating system.

License Type

Specify one of the following license types, based on the type of license you purchased:

- **Network License installation.** With this type of installation, you install the program to workstations with the files and registry entries that allow the program to communicate with the **Network License Manager**. You also define the configuration of the **Network License Manager** so that the licenses can be accessed. Workstations running the program based on a network installation do not require individual activation. Licensing of this program is managed by at least one license server.

The main advantage is that you can install Autodesk Navisworks Manage 2012 on more systems than the number of licenses you have purchased (for example, purchasing 25 licenses but installing on 40 workstations). At any one time, Autodesk Navisworks Manage 2012 runs on the maximum number of systems for which you have licenses. This means you get a true floating license.

- **Multi-Seat Stand-Alone installation (Stand-Alone option).** Choose this type of installation for stand-alone installations where a single serial number and product key are used for multiple seats. Multi-seat stand-alone installations do not rely upon a **Network License Manager** to manage product licenses; however, you can still use the Autodesk Navisworks Manage 2012 **Installation** wizard to create administrative images and create deployments. Registration and activation is more automated for multi-seat stand-alone installations. After the first activation using the multi-seat stand-alone serial number, activation occurs automatically for all workstations based on this deployment, as long as your systems are connected to the Internet.
- **Stand-Alone installation (Stand-Alone option).** Choose this type of installation for stand-alone installations where a single serial number

and product key are used for a single seat. Like a multi-seat stand-alone installation, you do not use the **Network License Manager** to manage product licensing, but installation, registration, and activation occurs on each workstation.

If you choose one of the stand-alone installation types, you can proceed to the section [Distribute the Program](#) (page 52).

Choose a License Server Model

If you chose the **Network License** option, you need to decide which license server model to use to distribute the product licenses.

TIP If you are deploying a stand-alone or multi-seat stand-alone installation type, you do not use a license server model. Proceed to the section [Distribute the Program](#) (page 52).

For the network installation, use one of the following license server models:

- **Single license server model.** The **Network License Manager** is installed on a single server, so license management and activity is restricted to this server. A single license file represents the total number of licenses available on the server.
- **Distributed license server model.** Licenses are distributed across more than one server. A unique license file is required for each server. To create a distributed license server, you must run the **Network License Manager** on each server that is part of the distributed server pool.
- **Redundant license server model.** You use three servers to authenticate a single license file. One server acts as the master, while the other two provide backup if the master server fails. With this configuration, licenses continue to be monitored and issued as long as at least two servers are still functional. The license file on all three servers is the same. You must install the **Network License Manager** on each server.

Each of these license server models is described in detail in the *Autodesk Licensing Guide*. It is strongly recommended that you read that guide before you deploy the program. You can find the *Autodesk Licensing Guide* by clicking the **Installation Help** link at the lower left corner of the Autodesk Navisworks Manage 2012 **Deployment** wizard.

Set Up Network Tools and Your License Server

If you are planning to have users run the program using network licenses, you need to use the **Network License Manager**.

The **Network License Manager** helps you configure and manage license servers.

Install the Network License Manager

The **Network License Manager** is used to configure and manage the license servers.

To install your Network License Manager

- 1 In the Autodesk Navisworks Manage 2012 **Installation** wizard, click **Install Tools and Utilities**.
- 2 On the **Configure Installation** page, select **Network License Manager** and click **Install**.

NOTE You can accept the default installation path (C:\Program Files\Autodesk) or **Browse** to specify a different path. If you enter a path that does not exist, a new folder is created using the name and location you provide.

WARNING Do not install the **Network License Manager** on a remote drive. When you install the **Network License Manager** files, you must provide a path to a local drive. You must specify the drive letter; the universal naming convention (UNC) is not supported.

- 3 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed with the installation. Choose your country or region, click **I Accept**, and then click **Next**.
- 4 When the **Installation Complete** page displays, click **Finish**.

To Activate a Network License Through Autodesk.com

If your product does not include the Network License Activation Utility, you can activate your license by going to <https://registeronce.autodesk.com>, and following the on-screen instructions.

Configure Your License Server

You configure a license server so that you can manage the Autodesk Navisworks product licenses.

You can configure the license server with the **lmtools.exe** utility.

You should be logged in with Administrator rights when working with the **LMTOOLS** utility.

To configure your license server

- 1 Do one of the following:
 - (Windows XP) Click **Start > All Programs > Autodesk > Network License Manager > LMTOOLS**.
 - (Windows Vista and Windows 7) Double-click the **LMTOOLS** icon on the desktop.
- 2 In the **Lmtools** program, on the **Service/License File** tab, select the **Configure Using Services** option.
- 3 Click the **Config Services** tab.
- 4 In the **Service Name** list, select the service name you want to use to manage licenses.

By default, the service name is *FLEXnet Service 1*. If FLEXnet® is managing other software on your computer in addition to Autodesk, you can change the service name to avoid confusion, for example, you can rename *FLEXnet Service 1* to *Autodesk Server1*.

NOTE If you have more than one software vendor using FLEXnet for license management, the **Service Name** list contains more than one option. Make sure that only one Autodesk service is listed.
- 5 In the **Path to Lmgrd.exe File** field, enter the path to the **Network License Manager** daemon (*lmgrd.exe*), or click **Browse** to locate the file.

By default, this daemon is installed in the *C:\Program Files\Autodesk Network License Manager* folder.
- 6 In the **Path to the License File** box, enter the path to your license file, or click **Browse** to locate the file.
- 7 In the **Path to the Debug Log File** box, enter a path to create a debug log, or click **Browse** to locate an existing log file.

It is recommended that you save to the *\Program Files\Autodesk Network License Manager* folder. The log file must have a *.log* file extension. For new log files, you must enter the *.log* extension manually.

- 8 To run *lmgrd.exe* as a service, select **Use Services**.
- 9 To automatically start *lmgrd.exe* when the system starts, select **Start Server at Power Up**.
- 10 Click **Save Service** to save the new configuration under the service name you selected in step 4. Click **Yes** when prompted if you would like to save the settings to the service.
- 11 Click the **Start/Stop/Reread** tab and do one of the following:
 - If a service has not yet been defined for Autodesk, click **Start Server** to start the license server.
 - If a service for Autodesk is already defined and running, click **ReRead License File** to refresh the **Network License Manager** with any changes made to the license file or **Options** file.

The license server starts running and is ready to respond to client requests.

- 12 Close *lmtools.exe*.

Distribute the Program

Once you have fully prepared for creating a deployment, you are ready to set up and distribute Autodesk Navisworks by using the **Deployment** wizard and choosing a deployment method.

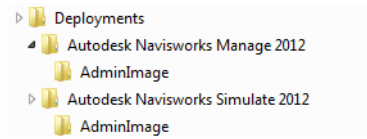
Create a Network Share

A network share is an installation folder that you make available to users' computers on a network. You point users to this location to install the program. Create a network share that will be used by the Autodesk Navisworks Deployment wizard during the creation of a client deployment.

A *shared folder* is required for network license and multi-seat stand-alone methods of installation.

It is recommended that you name the network share folder *Deployments* on the desktop of the system where you want deployments stored. You can then add subfolders inside the shared *Deployments* folder that clearly convey the

name of product you plan to deploy. This is also beneficial if you plan to deploy multiple products.



Any subfolders that are placed inside a shared folder are automatically shared.

TIP You must have **Full Control** permissions set for your shared folder when you are creating your deployment images. Read permissions are necessary to access the network share and administrative permissions on the workstation where the program is deployed.

To create your network share

- 1 On the desktop of a network server, create a folder named *Deployments*.
- 2 Right-click the *Deployments* folder and click **Share and Security** (or **Sharing**).
- 3 In the [folder name]**Properties** dialog box, **Sharing** tab, select **Share This Folder**.
- 4 Specify a **Share Name**, such as *Deployments*, if necessary.
- 5 Click the **Permissions** button. In the **Permissions** dialog box make sure **Full Control** is active. Click **OK**.
In Windows Vista and Windows 7, right-click the *Deployments* folder and then click **Share**. In the **Properties** dialog box select **Sharing** and then **Advanced Sharing** to share the folder. Click the **Permissions** button to make sure **Full Control** is active. Click **OK**.
These steps are important when creating your deployment images.
- 6 Click **OK** or **Close** to close the **Properties** dialog box.
- 7 For each product you plan to install, create a subfolder in the *Deployments* folder. Name each folder with the pertinent product name.

Minimize the Chances of Installation Failure

The Autodesk Navisworks installation process may stop if some applications, such as Microsoft Outlook or virus checking programs, are running when you

are creating a deployment. Close all running applications and temporarily disable virus checking utilities.

Distribute an Autodesk Navisworks Product

You can choose from several methods of distributing an Autodesk program. Network sharing is the default method.

- **Network Share.** Users launch the program with the shortcut icon that you created with the Deployment wizard. The program is installed on users' local computers, and a product icon appears on their desktop.

NOTE Users must have Read permissions to access the network share and administrative permissions on the workstation where this program is installed.

- **Scripting.** A script executes a command or a set of commands. Scripts are similar to batch files but are more flexible. Installation scripts are most useful for stand-alone installation of programs on computers that are connected to a network. Scripts are also useful for installing service packs, extensions, and object enablers on a network.

- **Imaging Software.** You can use imaging software, such as Norton Ghost, to create a master image to distribute Autodesk products. Once created, the master image is then replicated to other computers throughout your facility.

Care needs to be taken since the use of imaging software can result in conflicts with the product licensing, incomplete installations, and problems with activation.

Set Up a Deployment

The deployment process provides you with numerous options for creating, and customizing your deployments, so you should set aside ample time to complete the process in one sitting.

To be successful it is recommended that you review the following checklist and information before you begin. The deployment process is initiated from the Deployment wizard. Once a deployment is created, users then access the deployment to install products to their computers.

Preliminary Tasks for a Network Deployment

This checklist identifies preliminary tasks to complete, information to gather, and decisions to make before creating a network deployment.

TIP To review deployment settings, including details of product configuration, you can create a test deployment. After the learning process, you can delete the test deployment.

Deployment Checklist	
☐	Review the system requirements by clicking the System Requirements link in the installer. Confirm that your network, servers, and client workstations meet these system requirements.
☐	You understand the type of license you've purchased. If you plan a network license deployment, you should also be familiar with the type of license server model you want to use and the license server names.
☐	You have installed and activated any supporting tools and utilities. For example, if you are using a network license, install the Network License Manager from the installer.
☐	You have located your product serial number and product key. The serial number and product key are located on the outside of the product packaging, or in the email you received if you downloaded your product or upgrade from Autodesk.
☐	You have identified the location (such as a shared folder) where deployments will reside for each program you plan to deploy.
☐	You have closed all other programs and disabled anti-virus software.

Deployment Checklist	
☐	You have decided what languages you will include in your administrative image, and what languages you will use for your deployment package. NOTE Languages can be included when you initially create a deployment, or when you create a new deployment configuration, but not when you modify a deployment.
☐	Determine whether to create log files on individual computers, in the deployment folder, or both.
☐	Determine whether to run installations in silent mode. NOTE When products are installed in silent mode, the user's system can automatically reboot without warning when the installation is complete.
☐	Identify other applicable configuration settings such as locations of support files, and whether to automatically install service packs.
☐	You have decided if you want to include centralized product settings with your deployment, such as global options, workspaces, datatools, avatars, Clash Detective rules and custom tests, Presenter archives, object animation scripts, and so on.

Choose a Language

You can select a different language for deployment instructions, and a language for product installations in the same deployment process.

For example, if you need to provide Autodesk Navisworks to users in one or more languages different than your own, you can download language packs from the installer that will allow you to deploy the product in the languages you need to support.

NOTE Language packs can only be included in an administrative image during the creation of the deployment and not during modification.

When you start the deployment process, the installer automatically determines your operating system language. If a supported language is detected, your deployment pages are displayed in that language. If you want to change that language, you can choose a different one from the installer language list on the opening page of the Installation wizard.

NOTE Some products may not have multi-language support at the time of product release. Additional language support may be available later. Check <http://support.autodesk.com> for the availability of additional language packs.

Using Language Packs

Language packs support use of different languages in each Autodesk Navisworks product, including exporters. Pack names start with **NAVFREE_**, **NAVSIM_**, **NAVMAN_**, and **exporters_** respectively.

NOTE You can only select one language pack for each product for deployment.

It is possible to manually install additional language packs to Autodesk Navisworks products later by double-clicking on the language pack MSI file. Alternatively, you can set up and run scripts to install additional language packs.

Language packs are located on the installation DVD and unpacked downloaded media under the **x86** folder for 32-bit products and under the **x64** folder for 64-bit products.

- Language packs for specific products are included in the **NAVFREE**, **NAVSIM**, **NAVMAN**, and **NWEXPORT** subfolders of **x86** and **x64** folders.
- Language packs for specific languages are included in the **en-US** (English), **de-DE** (German), **es-ES** (Spanish), **fr-FR** (French), **it-IT** (Italian), **ja-JP** (Japanese), **ko-KR** (Korean), **pt-BR** (Brazilian Portuguese), **ru-RU** (Russian) and **zh-CN** (Chinese PRC) subfolders of the product folders.

Your Deployment Choices

When you create a deployment, you will make several choices during the process to create various client deployment images and deployment types. The following sections outline your choices in more detail.

What Is Silent Mode?

When silent mode is active and a user initiates the deployment, the installation proceeds without any explicit user input.

No dialog boxes are presented that require interaction from the user. This includes all error and warning dialog boxes. Check the log file for errors that may occur in the event of installation problems.

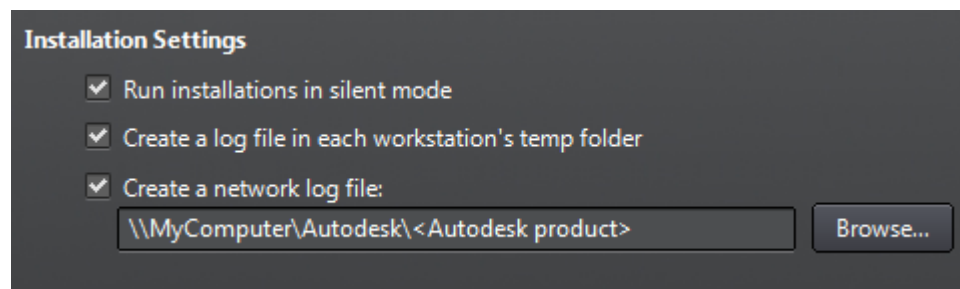
Enable silent mode by clicking the **Run installations in silent mode** check box in the **Configure** page.

To prevent a silent install from failing on client computers:

- Install Microsoft .Net Framework 4.0 on the client computers
- Reboot client computers before installing Autodesk Navisworks Manage 2012

Specify Log File Locations

On the **Create Deployment > Configure** page of the deployment process, you can choose whether or not to create a log file by selecting the appropriate check box in the **Installation Settings** area. The program has two types of log files with which you can monitor information about deployments and installations.



- **Network log.** The network log file keeps a record of all workstations that run the deployment. The log lists the user name, workstation name, and the status of the installation. Refer to this file for status information and details about problems that users encountered during installation (for example, low disc space or inadequate permissions).

The network log is named with the same name you chose for your deployment. You can specify where the log file is created by entering either

a valid UNC (universal naming convention) path or hard-coded path on your network, for example `\\MyComputer\Autodesk\<Autodesk product>`. Users should use their actual computer name in place of *MyComputer*.

NOTE The folder where the network log resides must be a shared folder where users who install the program have Change permissions. Otherwise, successes or failures for user installations cannot be written to the log file.

- **Client log.** The client log contains detailed installation information for each workstation. This information may be useful in diagnosing installation problems. The client log is located in the "*Temp*" directory of each client workstation.

To specify a log file location

- 1 While creating a deployment, on the **Configure Deployment** page, select the **Create a Network Log File** check box. Enter the name of the folder where you want the network log to be located.
- 2 If you want to create a client log, select the **Create a Log File in Each Workstation's Temp Folder** option.
- 3 Click **Next**.

Select a License Type

When you set up your deployment, you choose the type of installation to deploy based on the type of software license you purchased: stand-alone or network. You also select the network license server model you want to use to distribute product licenses.

IMPORTANT Depending on the type of option you choose, network or stand-alone license, note that you must use the same type of installation for all those products in your deployment session. If you select products that do not support the type of license you purchased, you will not be able to activate those products.

Specify the license type to use during deployment

- **Stand-Alone License** (a single serial number for a single seat). For a **Stand-Alone License**, you install, register, and activate the program on each workstation.
- **Network License.** With this type of installation, you install the program to workstations with the files and registry entries that allow the program

to communicate with the **Network License Manager**. You also define the configuration of the **Network License Manager** so that the licenses can be accessed. Workstations running the program based on a network installation do not require individual activation. Licensing of this program is managed by at least one license server.

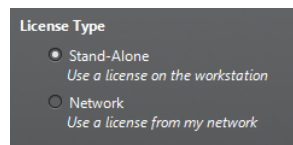
Specify the license server model during deployment

If you choose the **Network license** option, you need to decide which license server model to use to distribute your product licenses - single, distributed or redundant license server model.

If you are deploying a stand-alone or multi-seat stand-alone installation type, you do not use a license server model. For more information on server models, see [Choose a License Server Model](#) (page 49).

To deploy a Stand-Alone License

- 1 While creating a deployment, on the **Product Information** page, select **Stand-Alone**.



- 2 Click **Next**.

To deploy a network license using a single license server model

- 1 While creating a deployment, on the **Product Information** page, select the **Network** option.
- 2 Select **Single License Server** as the license server model you want to use with the **Network License Manager**.

If you have already used the **Network License Manager** to create a license server model, you must select the same license server model in this step. If you have not yet used the **Network License Manager** to create a license server model, make sure that you select the same settings in the **Network License Manager** that you choose here.

- 3 Enter the server name of the server that will run the **Network License Manager**, or click the **Browse** button to locate the server. Click **Next**.

For more information about license server models and setting up your license server, see [Choose a License Server Model](#) (page 49) or [Configure Your License Server](#) (page 51).

To deploy a network license using a distributed license server model

- 1 While creating a deployment, on the **Product Information** page, select the **Network** option.
- 2 Select **Distributed License Server** as the license server model you want to use with the **Network License Manager**.
If you have already used the **Network License Manager** to create a license server model, you must select the same license server model in this step. If you have not yet used the **Network License Manager** to create a license server model, make sure that you select the same settings in the **Network License Manager** that you choose here.
- 3 Enter the name of one of the servers that will run the **Network License Manager**, or click the **Browse** button to locate the server. Click **Add** to add the server to the **Server Pool**. Once all the servers are added to the **Server Pool** list, use the **Move Up** and **Move Down** buttons to arrange the servers in the order you want them to be searched by a user's workstation. You must enter at least two servers. Click **Next**.

For more information about license server models and setting up your license server, see [Choose a License Server Model](#) (page 49) or [Configure Your License Server](#) (page 51).

To deploy a network license using a redundant license server model

- 1 While creating a deployment, on the **Product Information** page, select the **Network** option.
- 2 Select **Redundant License Server** as the license server model you want to use with the **Network License Manager**.
If you have already used the **Network License Manager** to create a license server model, you must select the same license server model in this step. If you have not yet used the **Network License Manager** to create a license server model, make sure that you select the same settings in the **Network License Manager** that you choose here.
- 3 In the **First Server Name** field, enter a server name of one server that will run the **Network License Manager**, or click the **Browse** button to locate the server. Enter the server names for the remaining two servers that you will use in the redundant server pool. Click **Next**.

NOTE If you are not sure how to obtain the server host name, see **Plan Your License Server Configuration** in the *Autodesk Licensing Guide* by clicking the **Installation Help** link at the lower left corner of the **Deployment** wizard.

For more information about license server models and setting up your license server, see [Choose a License Server Model](#) (page 49) or [Configure Your License Server](#) (page 51).

Customize Deployment

When you create a deployment, you can accept the default settings for any product you choose to install, or you can configure these settings to create a custom deployment. To configure product settings, on the **Create Deployment** page of the installer, ensure that the product is selected for installation, and then click the product in the scrolling list. The configuration panel opens. Most products allow you to choose an installation type, either **Typical** to select the most common options, or **Custom** to review options in more detail. You can also select the **Project and Site Folders** and configure additional deployment settings.

After you have configured the settings as required, click the product name to close the configuration panel.

NOTE The configuration settings you choose apply to every instance of the product installed from the deployment. After the deployment is created, these settings can be changed only by modifying the deployment. For more information, see [Modify a Deployment](#) (page 69).

Configure Individual Products

As part of making your deployment choices, you can accept the default settings for any product you choose to install, or you can configure these settings to create a custom deployment.

- **Typical.** Installs the most common application features. This option is recommended for most users.
- **Custom.** Installs only the application features that you select.

To deploy a typical installation, and specify a product location

- 1 During the deployment, **Typical** is the default installation type. You can confirm this on the **Configure Deployment** page, by clicking a product name to open the configuration panel where you can view the installation type.
- 2 Enter the path on the client workstation where you want to install the program, for example *C:\Program Files\<Autodesk product>*.
- 3 Click the product name again to close the configuration panel.

To deploy a custom installation, choose optional features, and specify a product location

- 1 During the deployment, on the **Configure Deployment** page, click a product name to open the configuration panel.
- 2 Select the **Custom** option as the type of installation that you want.
- 3 From the **Select Features to Install** list, select the features that you want to install.

Your choices are as follows:

Redistributable ActiveX Control	Contains Autodesk Navisworks Redistributable ActiveX control.
API	Contains the Component Object Model interface for customizing and extending the Autodesk Navisworks functionality.
Example NWD files	Contains various feature sample files.
Batch Utility	Contains an add-in to run and schedule commands.
Program	Contains full set of Autodesk Navisworks files.
Sample RPCs	Contains several Rich Photorealistic Content files.

NOTE For Autodesk Navisworks exporter plugins, currently you cannot change which plugins are installed. By default, the exporter plugins for all third-party products installed on a client computer will be automatically enabled.

If, after making feature changes, you decide you want the original selections, click the **Restore Defaults**.

- 4 Click the product name to close the configuration panel.
- 5 Enter the path on the client workstation where you want to install the program, for example *C:\Program Files\<Autodesk product>*.

Select the Project and Site Folders (optional)

When you set up your deployment, you can share various Autodesk Navisworks settings across entire project site, or across a specific project group depending on the required level of granularity.

See also:

[How do I share the Autodesk Navisworks settings on a site and project basis?](#) (page 73)

To select the Project and Site folders

- 1 On the **Configure Installation** page, click the product name to open the configuration panel where you can review and change settings.
- 2 In the **Project Folder** box, click the **Browse** button to select the directory that contains the Autodesk Navisworks settings specific to a project group.
- 3 In the **Site Folder** box, click the **Browse** button to select the directory that contains the Autodesk Navisworks settings standard across the entire project site.

Select Additional Configuration Options

You can customize user preferences, include additional files and service packs, and specify **Communication Center** options and access to online resources for each of the products you choose to install. See [Additional Deployment Configuration Options](#) (page 79).

Create a Deployment

A deployment contains a Microsoft Transform (MST) file that modifies the default behavior of the installation program.

As you begin a deployment, you need to select the location of your administrative image, a deployment name, and if your target systems are 32- or 64-bit operating systems.

Create a Deployment Using Default Values

This is the fastest means of creating client deployment images.

To create a default deployment using default values

- 1 Close all running applications on your computer and start the **Installation** wizard.
- 2 On the **Installation** wizard, if required, select an alternate language for the **Installation** wizard from the **Installation Instructions** drop-down, and then click **Create Deployment**.
- 3 On the **Configure Deployment** page, **Administrative Image** section, specify the following data:
 - a In the **Deployment Configuration Name** field, enter a descriptive name for the deployment that identifies the user group, platform, or other attribute to differentiate this deployment from others. The name you enter here is the name of the shortcut users will access to install the product.
 - b In the **Administrative Image Path** field, enter the path to your existing shared network location, or use the **Browse** button to navigate to that location. This area is where you create and store your administrative image. Users install the program from this location.

NOTE If you do not know how to create a network share, see [Create a Network Share](#) (page 52).
 - c Under **Target Platform** select either 32-bit or 64-bit for your target operating system. This selection does not identify the system your deployment was created on; it identifies your target system.

d Choose whether to **Include Only the Products Required by this Configuration**.

- If selected, the deployment cannot be modified in future to include additional products.
- If unselected, the administrative image will include all possible products, so the current deployment can be modified in any way.

TIP It is recommended that you include all available products if you plan to create multiple deployments from this administrative image with different product mixes, and prefer not to use the installation media again. If there are products you will never use, or if you do not expect to create additional deployments, you should select a subset of products.

4 In the **Installation Settings** section, select one or more of these options:

- **Silent Mode.** The installation runs in background, without prompting the user for any input.

NOTE Silent mode can automatically restart the user's computer without warning after installation.

- **Workstation Log.** Log file on each workstation can enable users to review and troubleshoot their own installations.

- **Network Log.** Log file in a central location supports the network administrator in reviewing data for all installations.

NOTE The network log file must be in a shared folder where users of the deployment have read-write access. Otherwise, log data for user installations cannot be written to the log file. For more information regarding log files, refer to [Specify Log File Locations](#) (page 58).

5 Click **Next**.

6 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed with the installation. Choose your country or region, click **I Accept**, and then click **Next**.

NOTE If you do not agree to the terms of the license and want to terminate the installation, click **Cancel**.

7 On the **Product Information** page, select a [License Type](#) (page 59) and enter your **Serial Number** and **Product Key**. If the data is valid, green check marks appear. Click **Next**.

- 8 On the **Configure Deployment** page, select the products to include in the deployment, and if required add language pack(s) from the **Product Language** drop-down (see [Choose a Language](#) (page 33)).
- 9 Click **Create**.
After the deployment is created, the **Deployment Configuration Complete** page opens, presenting useful paths to the new deployment and a link for registering products online.
- 10 After you have finished using this screen, click **Finish**.

Create a Deployment Using Configured or Customized Values

If you plan on altering your deployment by removing features or restricting access to online resources you can create a configured or customized deployment.

To create a custom deployment

- 1 Close all running applications on your computer and start the **Installation** wizard.
- 2 On the **Installation** wizard, if required, select an alternate language for the **Installation** wizard from the **Installation Instructions** drop-down, and then click **Create Deployment**.
- 3 On the **Configure Deployment** page, **Administrative Image** section, specify the following data:
 - a In the **Deployment Configuration Name** field, enter a descriptive name for the deployment that identifies the user group, platform, or other attribute to differentiate this deployment from others. The name you enter here is the name of the shortcut users will access to install the product.
 - b In the **Administrative Image Path** field, enter the path to your existing shared network location, or use the **Browse** button to navigate to that location. This area is where you create and store your administrative image. Users install the program from this location.

NOTE If you do not know how to create a network share, see [Create a Network Share](#) (page 52).

- c Under **Target Platform:** select either 32-bit or 64-bit for your target operating system. This selection does not identify the system your deployment was created on; it identifies your target system.
- d Choose whether to **Include Only the Products Required by this Configuration**.
 - If selected, the deployment cannot be modified in future to include additional products.
 - If unselected, the administrative image will include all possible products, so the current deployment can be modified in any way.

TIP It is recommended that you include all available products if you plan to create multiple deployments from this administrative image with different product mixes, and prefer not to use the installation media again. If there are products you will never use, or if you do not expect to create additional deployments, you should select a subset of products.

- 4 In the **Installation Settings** section, select one or more of these options:

- **Silent Mode.** The installation runs in background, without prompting the user for any input.

NOTE Silent mode can automatically restart the user's computer without warning after installation.

- **Workstation Log.** Log file on each workstation can enable users to review and troubleshoot their own installations.

- **Network Log.** log file in a central location supports the network administrator in reviewing data for all installations.

NOTE The network log file must be in a shared folder where users of the deployment have read-write access. Otherwise, log data for user installations cannot be written to the log file. For more information regarding log files, refer to [Specify Log File Locations](#) (page 58).

- 5 Click **Next**.

- 6 Review the Autodesk software license agreement for your country or region. You must accept this agreement to proceed with the installation. Choose your country or region, click **I Accept**, and then click **Next**.

NOTE If you do not agree to the terms of the license and want to terminate the installation, click **Cancel**.

- 7 On the **Product Information** page, select a [License Type](#) (page 59) and enter your **Serial Number** and **Product Key**. If the data is valid, green check marks appear. Click **Next**.
- 8 On the **Configure Deployment** page, select the products to deploy, and if required add a language pack(s) from the **Product Language** drop-down (see [Choose a Language](#) (page 33)).
- 9 If you need to configure settings for a product, click the product name to open the configuration panel, where you can review and change settings. See [Customize Deployment](#) (page 62).

NOTE The configuration settings you choose apply to every instance of the product installed from the deployment. After the deployment is created, these settings can be changed only by modifying the deployment.

- 10 Click the product name to close the configuration panel.
- 11 Click **Create**.
After the deployment is created, the **Deployment Configuration Complete** page opens, presenting useful paths to the new deployment and a link for registering products online.
- 12 After you have finished using this screen, click **Finish**.

Modify a Deployment (optional)

After a deployment is created, it may be necessary to modify the deployment for some client workstations.

You can apply a patch or select various custom files that are not part of the base administrative image. You can also perform modifications such as changing the installation directory from drive C to drive D.

To modify a deployment

- 1 Open the shared network folder where you originally chose to place your product deployment.
- 2 In the **Tools** folder, double-click the **Create and Modify a Deployment** shortcut.
This re-opens the **Deployment** wizard.
- 3 Click through the deployment pages and make the necessary changes.
- 4 After all the modifications have been made, click **Create Deployment**.

Point Users to the Administrative Image

When you have completed the deployment process, you are ready to have users install the newly created or modified deployment.

You need to notify your users of the shortcut that was created in the administrative image. The shortcut is the same name that you chose when you [created a deployment](#) (page 65).

To point users to the administrative image

- The simplest method of notifying users how to install the deployment is to email them with instructions about using the shortcut. At a minimum, the instructions need to include the location of the deployment and instructions about double-clicking the shortcut to the deployment.

Uninstall an Autodesk Product

When you uninstall an Autodesk product, all components are removed in the process.

This means that even if you have previously added or removed components, or if you have reinstalled or repaired an Autodesk product, uninstalling removes all Autodesk product installation files from your system.

IMPORTANT Do not use registry cleaning programs or attempt to modify the registry entries yourself to uninstall an Autodesk product. Failure to follow the official uninstall procedure will result in the inability to re-install the software.

To uninstall the program

- 1 Do one of the following:
 - (Windows XP) Click **Start > Settings > Control Panel > Add or Remove Programs**.
 - (Windows Vista and Windows 7) Click **Start > Control Panel > Programs and Features**.
- 2 From the list of programs, click Autodesk Navisworks Manage 2012, and then click **Change/Remove** (Windows XP) or **Uninstall/Change** (Windows Vista and Windows 7).

The Autodesk Navisworks Manage 2012 **Installation** wizard re-opens in **Maintenance Mode**.

- 3 Click **Uninstall**.
- 4 On the **Uninstall** Autodesk Navisworks Manage 2012 page, click **Next** to remove Autodesk Navisworks from the system.
- 5 When informed that the product has been successfully uninstalled, click **Finish**.

NOTE Even though Autodesk Navisworks is removed from your system, the software license remains. If you reinstall Autodesk Navisworks at some future time, you will not have to register and re-activate the program.

Installation Troubleshooting

This section provides solutions to installation issues and answers to commonly asked questions that may arise while installing your products.

Additional troubleshooting information and support is also available at <http://support.autodesk.com>.

General Installation Issues

This section provides solutions to installation issues and answers to commonly asked questions that may arise while installing your products.

How can I check my graphics card driver to see if it needs to be updated?

It is recommended that you ensure your computer has the most current graphics card driver for the best possible display performance.

To identify your graphics card driver

- 1 Start Autodesk Navisworks Manage 2012.
- 2 In the **InfoCenter** box, click the down arrow next to the **Help** button
 ➤ **System Info**.
The Autodesk Navisworks Manage 2012 information dialog box opens.
- 3 Review the information about your system including the graphics card driver and driver version, and click **OK** to close the dialog.

To check the Web for an updated graphics card driver

- Use Windows Update. If a more recent graphics card driver is available, select it to have Windows Update download and install it.
- Search the graphics card manufacturer's website for the type of installed graphics card. If a more recent graphics card driver is available, install it following the instructions provided by the manufacturer.

To install an updated graphics card driver

- 1 Check the Web for to see if an updated driver is available.
 - Use Windows Update.
 - Search the graphics card manufacturer's website for the type of installed graphics card.
- 2 If a more recent graphics card driver is available, follow the instructions from the website to download and install it.

How do I switch my license from stand-alone to network or network to stand-alone?

If you simply entered the wrong license type by mistake, and are still running the installer, use the **Back** button to return to the **Product Information** page, and change the **License Type**.

If you want to change the license type for an installed product, contact your Autodesk reseller or license supplier to obtain the new license and serial number. Then uninstall your product and run a new install to change the license type and enter the new serial number.

When performing a Typical installation, what gets installed?

A **Typical** installation includes the following features:

Redistributable ActiveX Control

Contains Autodesk Navisworks Redistributable ActiveX control.

API	Contains the Component Object Model interface for customizing and extending the Autodesk Navisworks functionality.
Example NWD files	Contains various feature sample files.
Batch Utility	Contains an add-in to run and schedule commands.
Program	Contains full set of Autodesk Navisworks files.
Sample RPCs	Contains several Rich Photorealistic Content files.

Why should I specify the Project Folder and Site Folder?

You can share global Autodesk Navisworks settings, workspaces, datatools, avatars, Clash Detective rules and custom tests, Presenter archives, object animation scripts, and so on, with other users.

These settings can be shared across an entire project site, or across a specific project group depending on the required level of granularity.

Autodesk Navisworks examines the current user profile and the all users profile on the local machine, and then checks the settings in the **Project Directory** and the **Site Directory**. The files in the **Project Directory** take precedence.

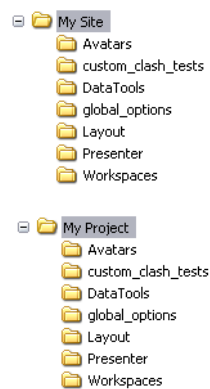
How do I share the Autodesk Navisworks settings on a site and project basis?

Sharing Autodesk Navisworks settings requires you to export the desired settings as an XML file to the appropriate Site or Project directory's *global_options* folder. The name of the XML file is not significant. However it must be stored in the *global_options* folder.

TIP When you configure global options, you can lock some of the options to prevent users from editing them later on local machines. To create a locked global options file, run the stand-alone **Options Editor** from the command line by typing “drive:pathname\OptionsEditor.exe” -l. The **Options Editor** opens with the locking facility.

To share settings on a site and project basis

- 1 Create appropriate Site and Project directories and subfolders in a central location to be accessed by other Navisworks users.



- 2 In Autodesk Navisworks, click the application button ► **Options Editor**.
- 3 Click **Export**.
- 4 In the **Select Options to Export** dialog box, check all options you want to export and click **OK**.
- 5 In the **Save As** dialog box, name the XML file as desired and save it to the *global_options* folder in the appropriate Site or Project directory.

See also:

[Location Options](#) (page 154)

[Autodesk Navisworks Options](#) (page 150)

[Select the Project and Site Folders \(optional\)](#) (page 64)

How do I change which exporter plugins are installed?

For Autodesk Navisworks exporter plugins, you can change which plugins are installed by clicking on an exporter plugin to open the configuration panel on the **Configure Deployment** page of the **Installation** wizard.

 Indicates the third-party software already installed on your computer.

 Indicates plugins for the third-party software, that are either not installed or have not been detected by the **Installation** wizard.

Select the check boxes next to all required plugins. If the third-party software has not been detected by the wizard, you can manually enable the plugins for it. Selecting the check box automatically opens the dialog box, which enables you to browse for the correct software installation directory.

How do I register and activate Autodesk Navisworks?

The first time you start Autodesk Navisworks Manage 2012, the **Product Activation** wizard is displayed. You can either activate Autodesk Navisworks at that time or run Autodesk Navisworks and activate it later.

Until you register and enter a valid activation code for Autodesk Navisworks Manage 2012, you are operating the program in trial mode and the **Product Activation** wizard is displayed for 30 days from the first time that you run the program. If after 30 days of running Autodesk Navisworks Manage 2012 in trial mode you have not registered and provided a valid activation code, your only option is to register and activate Autodesk Navisworks Manage 2012. You will not be able to run in trial mode after the 30 days expires. Once you register and activate Autodesk Navisworks Manage 2012, the **Product Activation** wizard is no longer displayed.

TIP The fastest and most reliable way to register and activate your product is by using the Internet. Simply enter your registration information and send it to Autodesk over the Internet. Once you submit your information, registration and activation occur almost instantly.

If you installed Autodesk Navisworks as part of a suite, a single trial period, usually 30 days, applies to all Autodesk products in the suite. When the first product is started, the trial period countdown begins for all products. If you

do not activate one of the products before the trial period ends, access to all of the products is denied. After the trial period ends, you can restore access to the products by entering an activation code.

To register and activate Autodesk Navisworks Manage 2012

- 1 Click **Start > All Programs > Autodesk > Autodesk Navisworks Manage 2012 > Autodesk Navisworks Manage 2012**.
- 2 In the Autodesk Navisworks Manage 2012 **Product Activation** wizard, select **Activate the Product**, and then click **Next**.
This starts the **Register Today** process.
- 3 Click **Register and Activate (Get an Activation Code)**.
- 4 Click **Next** and follow the on-screen instructions.

If you do not have Internet access, or if you want to use another method of registration, you can register and activate Autodesk Navisworks Manage 2012 in one of the following ways:

- **Email.** Create an email message with your registration information and send it to Autodesk.
- **Fax or Post/Mail.** Enter your registration information, and fax or mail the information to Autodesk.

Run the installer again from the original media, and click **Install Tools & Utilities** on the first screen. The installer guides you through the process of selection, configuration and installation of tools and utilities.

When should I reinstall the product instead of repairing it?

Reinstall your product if you accidentally delete or alter files that are required by the program.

Missing or altered files adversely affect the performance of your product and cause error messages when you try to execute a command or find a file.

If an attempt to repair an installation fails, reinstalling is the next best option.

When I uninstall my software, what files are left on my system?

If you uninstall the product, some files remain on your computer such as files you created or edited (for example, drawings or custom menus).

Your license file also stays on your computer when you uninstall your product. If you reinstall on the same computer, the license information remains valid; you do not have to reactivate the product.

Deployment Issues

This section outlines common issues and their solutions with regards to software deployments.

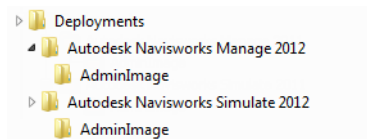
Is there a checklist I can refer to when performing a deployment?

The *Installation* chapter contains a complete section that describes preliminary actions and the entire deployment process. See [Preliminary Tasks for a Network Deployment](#) (page 55).

Where should deployments be located?

Shared folders are required for both network license and multi-seat stand-alone methods of installation.

The shared folder (*network share*) is created before you run the **Installation** wizard and is where product deployments are stored. It is recommended that you name the network share folder *Deployments* on the desktop of the system where you want deployments stored. You can then add subfolders inside the shared *Deployments* folder that clearly convey the names of products you plan to deploy. For example:



Any subfolders that are placed inside a shared folder are automatically shared.

NOTE You must have **Full Control** permissions set for your shared folder when you are creating your deployment images. Read permissions are necessary to access the network share and administrative permissions on the workstation where the program is deployed.

Where can I check if service packs are available for my software?

To find out if a patch or Service Pack is available for your product, visit the Autodesk Product Support page at <http://support.autodesk.com>.

How do I choose between 32-bit and 64-bit deployments?

You have a choice of selecting 32-bit or 64-bit deployment. This choice controls which operating system the deployment is targeted at, NOT what sort of products to include in the deployment.

- Selecting 32-bit deployment gives you a choice of 32-bit products, and will **only** install on 32-bit operating systems.
- Selecting 64-bit deployment gives you a choice of 32-bit and 64-bit products, and will **only** install on 64-bit operating systems.

So, for example, if you want to install available 32-bit products on a 64-bit operating system, you need to choose 64-bit deployment.

NOTE You can make either type of deployment from either type of operating system.

What are information channels?

The **Communication Center** allows you to receive announcements from various information channels. Through information channels, you can receive the following:

- Product Support information, including maintenance patch notifications.

- **Subscription Center** announcements and subscription program news, as well as links to e-Learning Lessons, if you are an Autodesk subscription member.
- Notifications of new articles and tips posted on Autodesk websites.

What are additional deployment configuration options?

The following options are available in the configuration panel in the **Configure Deployment** page of the **Installation** wizard.

User Preferences

- **Require Internet Explorer for Installation** (does not apply to Autodesk Navisworks)
- **Set DWfx as the Default Publishing Format** (does not apply to Autodesk Navisworks)
- **Set the Default Profile Name** (does not apply to Autodesk Navisworks)
- **Create a Desktop Shortcut for Autodesk Navisworks Manage 2012**

Service Packs

- **Include Service Pack(s) from Local Drive or Local Network**
- **Do Not Include Service Pack(s)**

TIP See [Where can I check if service packs are available for my software](#) (page 78).

Communication Center Options

- **Enable Live Updates**
- **Enable CAD Manager Channel** (does not apply to Autodesk Navisworks)
- **Enable RSS Feeds**
- **Allow User to Add RSS Feed**

Access to Online Resources

- **Specify the Ability to Access Online Tools** (does not apply to Autodesk Navisworks)
- **Specify the Customer Error (CER) Settings**
- **Specify the InfoCenter Search Settings** (does not apply to Autodesk Navisworks)

Licensing Issues

This section outlines common issues and their solutions with regards to software licenses and licensing your products.

What is the difference between a stand-alone license and a network license?

Stand-alone licensed products are registered and activated to an individual workstation. While the software can be installed on multiple systems in your facility, the license only allows one system to be operational. The **Portable License Utility** can be used if a license needs to be transferred to another system. If you need to run more systems, you need to purchase more stand-alone licensed products, or consider converting to network licenses.

Network licensed products rely on the **Network License Manager** to keep track of software licenses. The software can be installed and run on multiple systems, up to the maximum number of licenses you've purchased. The **Network License Manager** "checks out" licenses until they are all in use. No further systems can run the program until a license is "checked in". One main advantage of a network license is that you can install products on more computers than the number of licenses you have purchased. For example, you can purchase 25 licenses but install them on 40 computers for ease of access. At any one time, products can run on the maximum number of computers for which you have licenses. This means you get a true floating license. A network licensing system can also be configured to allow users to borrow a license for a limited time to use on a computer disconnected from the network.

What is the benefit to using a network licensed version of the software?

Network licensed products are recommended for large drafting/design facilities, classrooms, and lab environments.

The main advantage is that you can install products on more systems than the number of licenses you have purchased (for example, purchasing 25 licenses but installing on 40 workstations). At any one time, products will run on the maximum number of systems for which you have licenses. This means you get a true floating license. If software needs to be run on more systems, additional licenses can be purchased.

Registration and activation occurs only once and the licenses are maintained on your **Network License Server**.

What is Internet Explorer used for?

When you choose to activate the software, Internet Explorer makes this process much faster.

After installing your product, you can operate in *trial mode* for a given number of days. Whenever you launch the program, you are prompted to activate the software.

Once you entered your registration data and submit it to Autodesk, an activation code is returned and you are not prompted again during startup.

Networking Issues

This section outlines common issues and their solutions with regards to performing a network installation or configuring your network license servers.

Where do I find my server name?

When installing a network licensed product, you must specify the name of the server that will run the **Network License Manager**.

If you don't know the server name, you can quickly find it by opening a Windows command prompt on the system that will be the **Network License**

Manager. At the prompt, enter **ipconfig /all** and note the **Host Name** entry.

If I choose to create a log file, what kind of information does the log file contain?

There are two types of log files that can be generated that monitor information about deployments and installations.

- The Network log file keeps a record of all workstations that run the deployment. The log lists the user name, workstation name, and the status of the installation. Refer to this file for status information and details about problems that users may have encountered during installation.
- The Client log file contains detailed installation information for each workstation. This information may be useful in diagnosing installation problems. The client log is located in the \Temp directory of each client workstation.

What is an administrative image (MSI) file?

An *administrative image* is a collection of shared file resources created during the deployment process and is used by deployments to install the program to networked workstations. An MSI file is a Microsoft Installer file.

What is the impact of selecting all products to be included in the administrative image?

If you elect to include all products in your deployment, the administrative image will be larger.

You should select all products only when you create multiple deployments from this image and prefer not to use the installation DVD. If there are products you rarely or never use, and you do not expect to create additional deployments, you should only select a subset of products.

You can still create a deployment at a later date, and include additional products, but you need to create a new administrative image. You need the installation media to do so.

How should I configure a network license server for a firewall

If you have a firewall between the license server(s) and the client computers, you need to open some TCP/IP ports on the server for license-related communication. Open port 2080 for adskflex, and open ports 27000 through 27009 for lmgrd.

Uninstall and Maintenance Issues

This section outlines common issues and their solutions with regards to adding and removing features, reinstalling or repairing your installation, and uninstalling products.

IMPORTANT Do not use registry cleaning programs or attempt to modify the registry entries yourself to uninstall Autodesk Navisworks Manage 2012. Failure to follow the official uninstall procedure will result in the inability to re-install the software.

When adding or removing features, how can I tell what features get installed by default?

To quickly see what gets installed during a typical, default installation, click the **Restore Defaults** button on the **Add/Remove Features** page.

Is it possible to change the installation folder when adding or removing features?

Once your product is installed, you cannot change the installation path from the **Add/Remove Features** page. Changing the path while adding features results in program corruption, so it is not an option.

When should I reinstall the product instead of a repair?

You should reinstall your product if you accidentally delete or alter files that are required by the program. Missing or altered files adversely affect the performance of your product and cause error messages when you try to execute a command or find a file.

If an attempt to repair an installation fails, reinstalling is the next best option.

Installation data is cached locally on your drive and that data is reused when reinstalling. If any files cannot be located when reinstalling a product, you are prompted to load the original media. If the product was installed from a network deployment, you need access to the original deployment, unaltered by later changes such as the addition of a service pack.

When I uninstall my software, what files are left on my system?

If you uninstall the product, some files remain on your system such as files you created or edited.

Your license file also stays on your workstation when you uninstall your product. If you reinstall on the same workstation, the license information remains valid and you do not have to reactivate the product.

Quick Start

3

Start and Quit Autodesk Navisworks

Once you've [installed](#) (page 29) Autodesk Navisworks Manage 2012, you can start it from the Windows desktop or from the command line.

To start Autodesk Navisworks, do one of the following from the Windows desktop:

- Double-click the Autodesk Navisworks icon, or
- Click **Start** ► **All Programs** ► **Autodesk** ► Navisworks Manage 2012 ► Manage 2012.

Autodesk Navisworks starts in the language that best matches the settings on your computer. You can also start Autodesk Navisworks in [another of the installed languages](#) (page 40).

NOTE You can add command line switches to specify different startup routines for the program. See [Command Line Options](#) (page 88).

To quit Autodesk Navisworks, click the application button . At the bottom of the application menu, click Exit Autodesk Navisworks.

If no changes were made to the current project, the project closes and Autodesk Navisworks exits. If changes were made to the current project, you are prompted to save the changes. To save the changes to the project, click Yes. To continue exiting and discard the changes, click No. To return to Autodesk Navisworks, click Cancel.

Automatically Save and Recover Autodesk Navisworks Files

Power cuts, system and software failures can cause Autodesk Navisworks to close before you can save changes to your file.

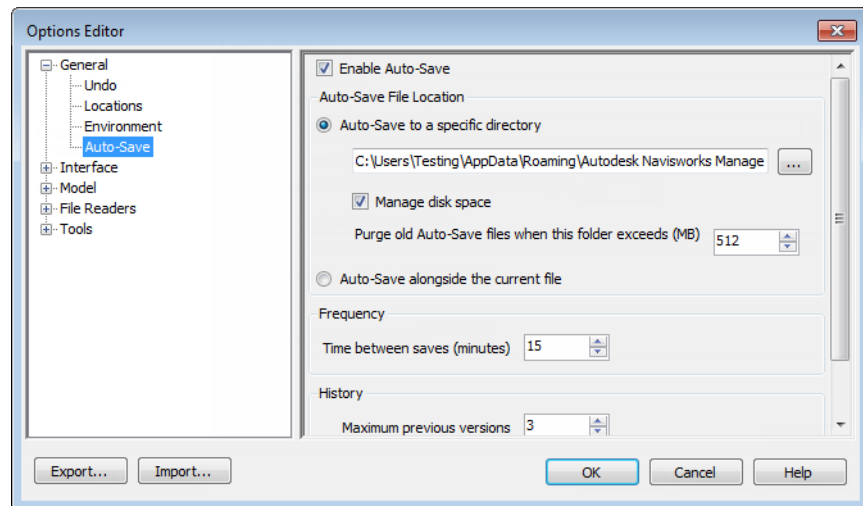
Autodesk Navisworks can automatically save backup versions of a file you are working on, enabling you to recover your work if Autodesk Navisworks closes abnormally.

Auto-saved files have an .nwf extension, and named as *<FileName>.AutoSave<x>* where **<FileName>** is the name of the current Autodesk Navisworks file, and **<x>** is a number that increments with each auto-save. So, for example, if you work with a file called *Enviro-Dome.nwd*, the first auto-saved file is called *Enviro-Dome.Autosave0.nwf*, the second auto-saved file is called *Enviro-Dome.Autosave1.nwf* and so on.

You can control a number of **Auto-Save** options, such as how often Autodesk Navisworks saves your work, the location of backup files, and the maximum number of backup files you want to keep.

To customize the Auto-Save options

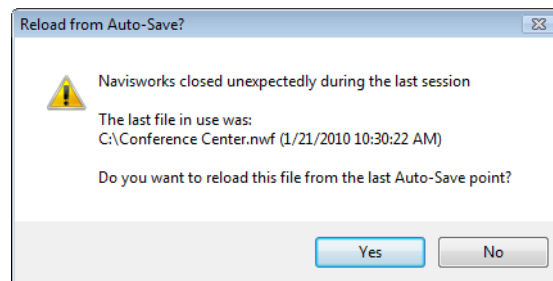
- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **General** node, and click **Auto-Save**.



- 3 Adjust the **Auto-Save** options, as required. For example, if you want Autodesk Navisworks to save a backup file every 20 minutes from a significant file change, enter **20** into the **Time Between Saves (minutes)** box.
- 4 Click **OK**.

To recover your work

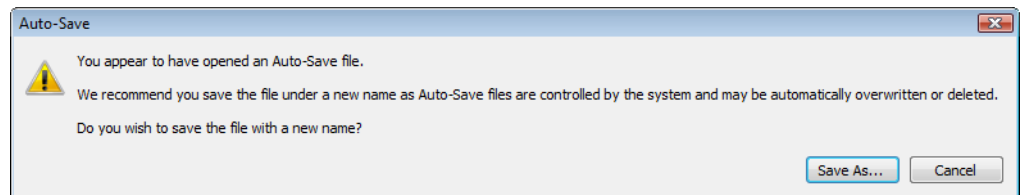
- 1 Start Autodesk Navisworks. You are automatically prompted to reload the last file you were working on.
- 2 Click **Yes** to open the most recent saved version of the file.



NOTE Click **No**, if you don't want to recover your work, or if you want to manually load a different backup file.

To manually load a backup file into Autodesk Navisworks

- 1 Start Autodesk Navisworks. If you are prompted to reload the last file you were working on, click **No**.
- 2 Click the application button ► **Open** ► **Open** .
- 3 In the **Open** dialog box, browse to the folder that contains the backup files. By default, it's <USERPROFILE>\ *Application Data* \<PRODUCTFOLDER>\ *AutoSave*.
- 4 Click **Open**.
- 5 When you are prompted to save a file with a different name, click **Save As**.



- 6 In the **Save As** dialog box, enter a new file name, and browse to the desired location.
- 7 Click **Save**.

 **Menu:** Classic user interface: **File** ► **Open**

 **Command entry:** CTRL + O

 **Toolbar:** Classic user interface: **Standard**, **Quick Access** toolbar 

Command Line Options

Command line switches can specify different startup routines for Autodesk Navisworks.

You can use command line switches to specify several options when you start the program. For example, you can run Autodesk Navisworks in another language, perform additional memory checks, load and append files, output error reports. With command line switches, you can also set up several program icons, each with different start-up options.

Command line switches are parameters you can add to the *roamer.exe* command line associated with a Microsoft® Windows® shortcut icon or the Windows

Run dialog box. You can include several switches within a single command line. Valid switches are listed in the following table.

Command Line Switch	Argument	Description
-dump	[file_name.dmp]	Outputs an error report to the specified file. You must provide the file name and file path within quotation marks.
-lang	en-US = English	Launches Autodesk Navisworks in the specified language. It is possible to run Autodesk Navisworks in a language other than the default locale (for example, you can run a Japanese version of Autodesk Navisworks with English default locale).
	de-DE = German	
	es-ES = Spanish	
	fr-FR = French	
	it-IT = Italian	
	ja-JP = Japanese	
	ko-KR = Korean	
	pt-BR = Brazilian Portuguese	
	ru-RU = Russian	
	zh-CN = Chinese (PRC)	
-log	[filename.txt]	Outputs log to the specified file. You must provide the full file path within quotation marks. For example, "C:\temp\log.txt"

Command Line Switch	Argument	Description
-memcheck		Runs Autodesk Navisworks with additional checks for memory errors.
-nwc	[inputfile]	Autodesk Navisworks converts an input file into the NWC format in the background. You must provide the full file path within quotation marks. For example, "C:\temp\a.dwg"
-nwd	[outputfile.nwd] [input file]	Autodesk Navisworks converts an input file into the NWD format in the background, and places it into the specified directory. You must provide the full file paths within quotation marks. For example, "C:\temp\b.dwg"
-options	[filename.xml]	Starts Autodesk Navisworks, and imports the global options from the specified global options file. You must provide the full file path within quotation marks. For example, "C:\temp\my_global_options.xml"
-regserver		Registers roamer.exe with COM.

The syntax for using command line switches is:

"drive:pathname\roamer.exe" [switches] ["file1"] ["file2"] ["fileN"],
 where [switches] are the valid command line switches in any order, and
 [file1]...[fileN] are the files to be loaded and appended together, if required.
 You must provide the full file paths within quotation marks.

For example, the following entry starts the program from a folder named *Autodesk Navisworks* in *Russian* language, loads the global options file *options.xml*, and creates a log file *log.txt*.

```
"C:\Autodesk Navisworks\roamer.exe" -options "C:\temp\options.xml"  
-lang ru-ru -log "C:\temp\file.log"
```

To start the program with a command line switch

- 1 Right-click the program icon on the Windows desktop. Click **Properties**.
- 2 In the Autodesk Navisworks Properties dialog box, **Shortcut** tab, in the **Target** box, edit the parameters for the switch using the following syntax:

```
"drive:pathname\roamer.exe" [switches] ["file"] ["file2"]  
["fileN"], where [switches] are the valid command line switches in any  
order, and [file1]...[fileN] are the files to be loaded and appended together,  
if required. You must provide the full file paths within quotation marks.
```

For example, enter: **"D:\Autodesk Navisworks\roamer.exe" -log
"D:\temp\nw_log.txt" "D:\temp\a.nwd" "D:\temp\b.dwg"**.

- 3 Click **OK**.

The User Interface

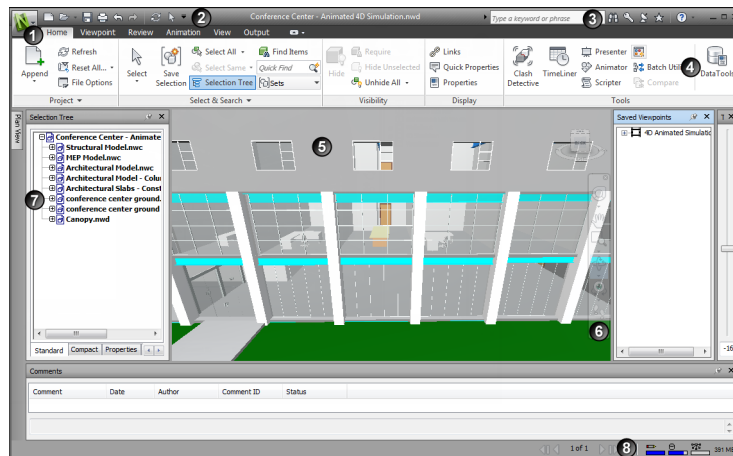
The Autodesk Navisworks interface contains a number of traditional Windows elements, such as the application menu, Quick Access toolbar, ribbon, dockable windows, dialog boxes and shortcut menus in which you complete tasks.

Parts of Autodesk Navisworks Interface

This section briefly describes the main components of the standard Autodesk Navisworks interface.

The Autodesk Navisworks interface is intuitive and easy to learn and use. You can adjust the application interface to match the way you work. For example, you can hide docking windows that you rarely use, so they do not clutter the interface. You can add and remove buttons from the ribbon and the **Quick Access** toolbar.

You can apply a different theme to the standard interface. You can also switch back to the classic Autodesk Navisworks interface with old-style menu and toolbars.



1. Application button and menu	5. Scene View
2. Quick Access toolbar	6. Navigation bar
3. InfoCenter	7. Dockable windows
4. Ribbon	8. Status bar

See also:

[Overview of InfoCenter](#) (page 11)

To change theme of the standard user interface

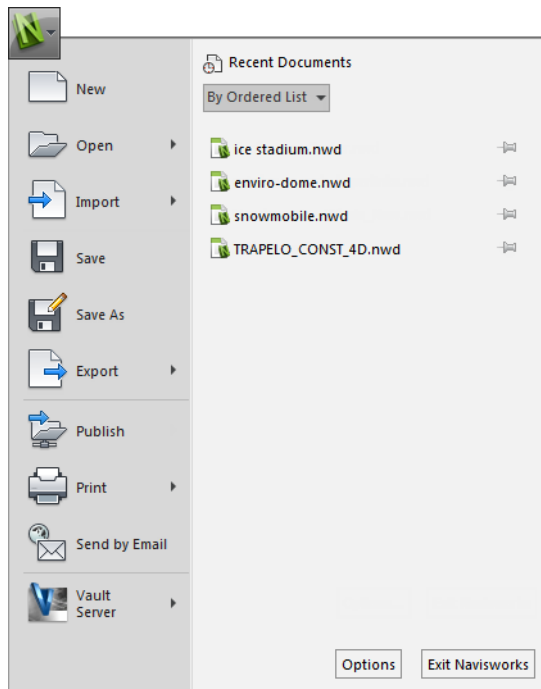
- 1 Click the application button  ➤ **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **User Interface** option.
- 3 On the **User Interface** page, select the required theme type from the **Theme** drop-down list.
- 4 Click **OK**.

Application Button and Menu

The application menu enables you to access common tools.

It provides access to many common file actions, and also allows you to manage your files using more advanced tools, such as Import, Export, and Publish. Some application menu choices have additional menus that show related commands.

To open the application menu, click the application button . Clicking it again closes the application menu.



Option	Description
 (New)	Closes the currently open file, and creates a new file.
 (Open)	Opens files.

Option	Description
 (Import)	Imports data into the current project.
 (Save)	Saves the current file.
 (Save As)	Saves your project as one of the native Autodesk Navisworks formats (NWF or NWD).
 (Export)	Exports data from the current project.
 (Publish)	Publishes the current project.
 (Print)	Prints the scene and sets print-related settings.
 (Send by Email)	Creates a new email with the current file as an attachment.
 (Vault Server)	Launches the Vault standalone client and enables you to login and logout of the Vault server, and check files in and out. By default, this option is not shown. To turn it on, use the Options Editor (Tools > Vault > Show in User Interface) .
Options	Opens the Options Editor.
Exit Navisworks	Exits the program.

Recent Documents List

You can view, sort, and access supported files that you have recently opened.

The most recent files are shown in the **Recent Documents** list. The list is ordered with the most recently used file at the top.

By default, up to four files are shown. If you want to modify the size of this list, use the **Options Editor**.

You can pin the files by using the push pin button to the right. Pinning enables you to keep a file in the list until you turn off the push pin button.

Sort and Group Files

Use the drop-down list at the top of the **Recent Documents** list to sort or group files by:

- **By Ordered List**
- **By Access Date**
- **By Size**
- **By Type**

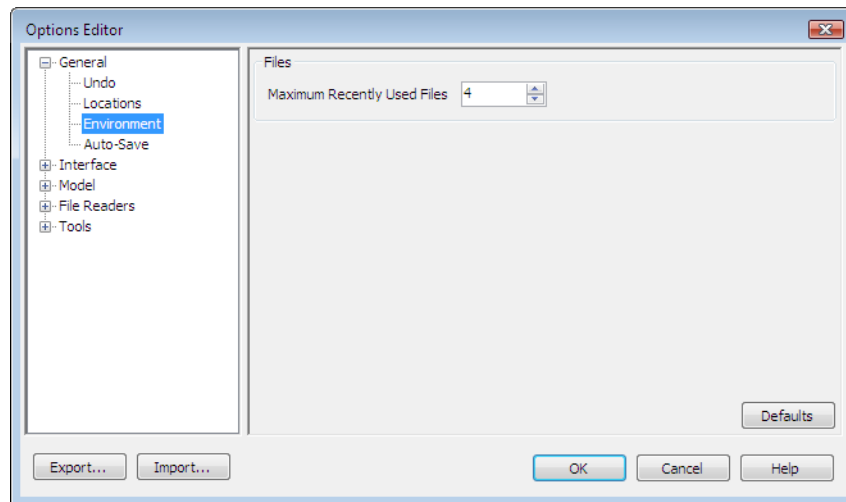
Preview Documents

When you mouse over a file in the **Recent Documents** list, the following information is displayed:

- Path where the file is stored
- Date the file was last modified
- Name of the person who is currently working with the file

To change the number of recent documents listed

- 1 Click the application button  > **Options**.
- 2 In the **Options Editor**, expand the **General** node, and click the **Environment** option.
- 3 On the **Environment** page, enter the number of recent documents to be listed into the **Maximum Recently Used Files** box.



4 Click **OK**.

To keep a document in the Recent Documents list

- Click the push pin button to the right of the document.

To view the Recent Documents list by access date

- In the top-left corner of the **Recent Documents** list, in the **By Ordered List** drop-down list, select **By Access Date**.

To view the Recent Documents list by size

- In the top-left corner of the **Recent Documents** list, in the **By Ordered List** drop-down list, select **By Size**.

To view the Recent Documents list by type

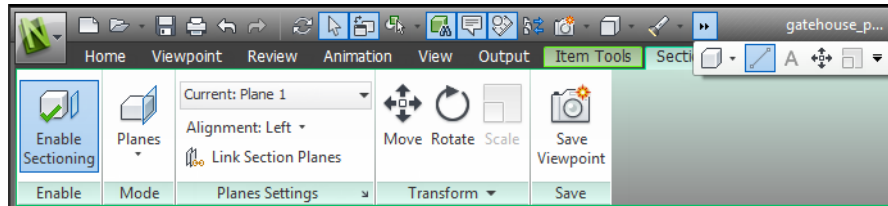
- In the top-left corner of the **Recent Documents** list, in the **By Ordered List** drop-down list, select **By Type**.

Quick Access Toolbar

At the top of the application window, the **Quick Access** toolbar displays frequently used commands.



You can add unlimited number of buttons to the **Quick Access** toolbar. Buttons are added to the right of the default commands. You can add separators between the buttons. Commands that extend past the maximum length of the toolbar are displayed in a flyout button  .



NOTE Only ribbon commands can be added to the **Quick Access** toolbar.

You can move the **Quick Access** toolbar either above or below the ribbon.

To add a ribbon button to the Quick Access toolbar

- 1 Display the tab and panel that contains the button you want to add to the **Quick Access** toolbar.
- 2 Right-click the button on the ribbon, and click **Add to Quick Access Toolbar**.

To remove a ribbon button from the Quick Access toolbar

- 1 Right-click the button on the **Quick Access** toolbar
- 2 Click **Remove from Quick Access toolbar**.

To display the Quick Access toolbar below the ribbon

- Click the **Customize Quick Access Toolbar** drop-down button, and click **Show Below the Ribbon**.

Shortcut menu: Right-click any button on the **Quick Access** toolbar. Click **Show Quick Access Toolbar below the Ribbon**.

To display the Quick Access toolbar above the ribbon

- Click the **Customize Quick Access Toolbar** drop-down button, and click **Show Above the Ribbon**.

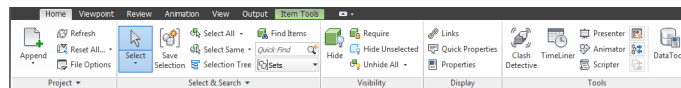
Shortcut menu: Right-click any button on the **Quick Access** toolbar. Click **Show Quick Access Toolbar above the Ribbon**.

By default, it contains the following tools:

Option	Description
 (New)	Closes the currently open file, and creates a new file.
 (Open)	Opens files.
 (Save)	Saves the current file.
 (Print)	Prints the current viewpoint.
 (Refresh)	Refreshes the files in the project.
 (Undo)	Cancels the most recent action.
 (Redo)	Reinstates the most recent action.
 (Select)	Selects items with a mouse click.
 (Customize Quick Access Toolbar)	Customizes the items displayed on the Quick Access toolbar. To enable or disable an item, click next to it on the Customize Quick Access Toolbar drop-down.

Ribbon

The ribbon is a palette that displays task-based tools and controls.



The ribbon is divided into tabs, with each tab supporting a specific activity. Within each tab, tools are grouped together into a task-based series of panels.

To specify which ribbon tabs and panels are displayed, right-click the ribbon and, on the shortcut menu, click or clear the names of tabs or panels.

You can customize the ribbon depending on your needs in the following ways:

- Change the order of ribbon tabs. Click the tab you want to move, drag it to the desired position, and release.
- Change the order of ribbon panels in a tab. Click the panel you want to move, drag it to the desired position, and release.

You can control the amount of space the ribbon takes in the application window. There are two buttons to the right of the ribbon tabs, that allow you to choose the ribbon toggle and ribbon minimize states.

- The first button toggles between the full ribbon state  and the minimize ribbon state .
- The second drop-down button  allows you to select one of four minimize ribbon states:
 - **Minimize to Tabs:** Minimizes the ribbon so that only tab titles are displayed.
 - **Minimize to Panel Titles:** Minimizes the ribbon so that only tab and panel titles are displayed.
 - **Minimize to Panel Buttons:** Minimizes the ribbon so that only tab titles and panel buttons are displayed.
 - **Cycle Through All:** Cycles through all four ribbon states in the order, full ribbon, minimize to panel buttons, minimize to panel titles, minimize to tabs.

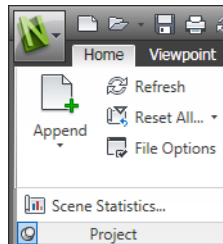
Contextual Tabs

Some of the tabs are contextual. When you execute some commands, a special contextual ribbon tab is displayed instead of a toolbar or dialog box. For example, as soon as you start selecting items in the **Scene View**, the previously hidden **Item Tools** tab appears. When nothing is selected, it becomes hidden again.

Slideout Panels

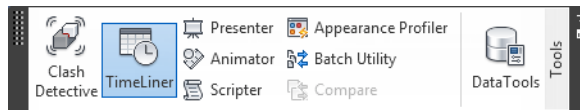
A down arrow ▼ to the right of a panel title indicates that you can slide out the panel to display additional tools and controls. By default, an expanded panel closes automatically when you click another panel. To keep a panel

expanded, click the push pin  icon in the bottom-left corner of the slideout panel.



Floating Panels

If you pull a panel off of a ribbon tab and into a different area in the application window or the desktop, that panel floats where you placed it. The floating panel remains open until you return it to the ribbon, even if you switch ribbon tabs.



Tool Launcher

Some ribbon panels display a dialog box or a dockable window related to that panel. A tool launcher arrow  in the lower-right corner of the panel indicates that you can display a related tool. Click the icon to display the associated dialog box or dockable window.

Check Boxes

Check boxes allow you to toggle an option on or off.

Sliders

When an option can be executed with varying intensity, the slider allows you to control the setting from lower to higher, or reverse.

To display the ribbon

If you use the **Classic** user interface, you can switch back to the ribbon.

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **User Interface** option.
- 3 On the **User Interface** page, select **Standard (Recommended)** from the **User Interface** drop-down list.
- 4 Click **OK**.

To hide or show a ribbon tab

- 1 Right-click anywhere inside the ribbon.
- 2 Under **Show Tabs**, click or clear the name of a tab.

To hide or show a ribbon panel

- 1 Click the ribbon tab that you want to organize.
- 2 Right-click the ribbon tab.
- 3 Under **Show Panels**, click or clear the name of a panel.

NOTE By default, the **Collaborate** panel on the **Review** tab is hidden.

To show or hide text labels on ribbon panels

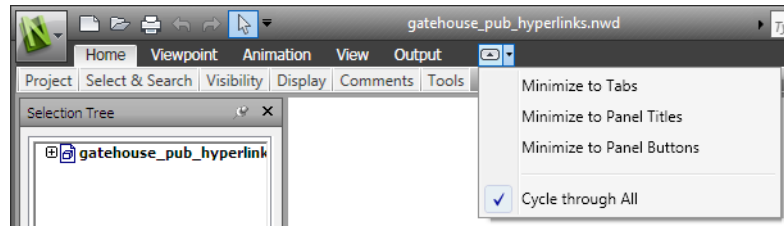
- Right-click a ribbon tab, and click or clear **Show Panel Titles**.

To return a floating panel to the ribbon

- Mouse over the right side of the floating panel and click the **Return Panels to Ribbon** icon.

To toggle the ribbon size

- 1 Click the drop-down arrow in the ribbon tab bar, and select the desired minimize ribbon state.



- 2 Double-click the name of the active ribbon tab or anywhere in the ribbon tab bar.

The ribbon toggles between the selected minimize ribbon state and the full ribbon state.

To reset the ribbon and the Quick Access toolbar

- 1 Right-click anywhere inside the ribbon.
- 2 Click **Restore Default Ribbon**.

Home Tab

Panel	Contains tools to...
Project	control the whole scene including appending files and refreshing CAD files, resetting changes made in Autodesk Navisworks, and setting file options.
Select & Search	select items and save selections in the scene via a range of methods, including using searches.
Visibility	show and hide items of model geometry.
Display	show and hide information including properties and links.
Tools	launch Autodesk Navisworks simulation and analysis tools.

Viewpoint Tab

Panel	Contains tools to...
Save, Load & Playback	save, record, load and playback saved viewpoints and viewpoint animations.
Camera	apply various settings to the camera.
Navigate	set the linear and angular speed of motion, select navigational tools and 3D mouse settings, and apply realism settings such as gravity and collisions.
Render Style	control the lighting and rendering settings.
Sectioning	enable cross- sectioning of the viewpoint in a 3D workspace.

Review Tab

Panel	Contains tools to...
Measure	measure distances, angles, and areas.
Redline	draw redline markup on the current viewpoint.
Tags	add and locate tags in the scene.
Comments	view and locate comments in the scene.
Collaborate	connect with other Autodesk Navisworks users across a network connection. This panel is hidden by default.

Animation Tab

Panel	Contains tools to...
Create	create object animations using the Animator tool or record viewpoint animations.
Playback	select and play back animations.
Script	enable scripts, or create new ones in the Scripter tool.

View Tab

Panel	Contains tools to...
Stereo	enable stereoscopic vision provided suitable hardware is available.
Navigate	set the linear and angular speed of motion, select navigational tools and 3D mouse settings, and apply realism settings such as gravity and collisions. This panel is hidden by default.
Navigation Aids	toggle navigation controls, such as the Navigation Bar, ViewCube, HUD elements, and reference views.
Scene View	control the Scene View window including going full screen, splitting the window and setting the background style/colors.
Workspace	control which floating windows are shown, and load/save workspace configurations.

Output Tab

Panel	Contains tools to...
Print	print and preview the current viewpoint, and set print settings.
Send	send an email with the current file as an attachment.
Publish	publish the current scene as an NWD file.
Export Scene	publish the current scene as a 3D DWF/DWFX, FBX, or Google Earth file.
Visuals	output images and animations.
Export Data	export data from Autodesk Navisworks, including Clash, TimeLiner, search and viewpoint data and PDS tags.

Item Tools Tab

Panel	Contains tools to...
SwitchBack	switch back to compatible design applications at the current view.
Hold	hold the selected items so that they move with you as you navigate around the scene.
Look At	focus and zoom the current view onto the selected items.
Visibility	control the visibility of the selected items.
Transform	move, rotate, and scale the selected items.

Panel	Contains tools to...
Appearance	change the color and transparency of the selected items.
Links	manage the links attached to the selected items.

Sectioning Tools Tab

NOTE Only available in a 3D workspace.

Panel	Contains tools to...
Enable	enable/disable sectioning of the current viewpoint.
Mode	switch the sectioning mode between planes and box modes.
Planes Settings	control section planes.
Transform	move, rotate, and scale the section planes/box.
Save	save the current viewpoint.

Vault

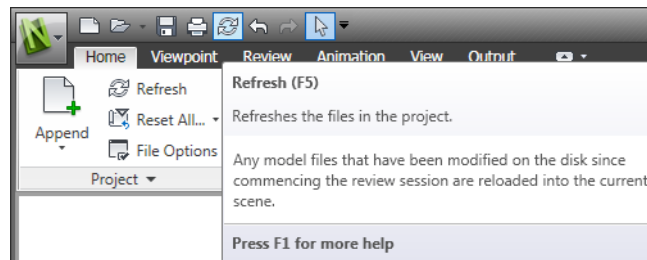
Panel	Contains tools to...
Access	open Vault and log in, log out, and append files.
File Status	refresh files and check them in and out.

Tooltips

Placing the mouse pointer over a menu choice or a button shows a tooltip containing the name of the tool, a keyboard shortcut (if applicable), and a brief description of the tool.

Some tooltips on the application menu, **Quick Access** toolbar, and ribbon are progressive. If you leave the cursor over the menu choice or a button for another moment, the tooltip may expand to show additional information.

While the tooltip is visible, you can press F1 for context-sensitive help that provides more information about that tool.

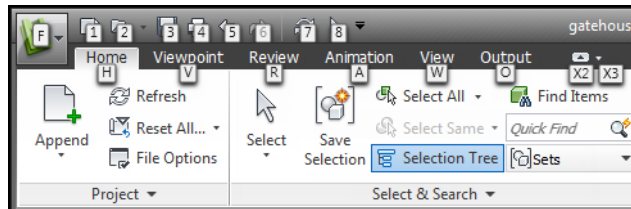


Keytips

Autodesk Navisworks provides accelerator keys, or keytips, to enable you to use the keyboard, rather than the mouse, to interact with the application window.

Keytips are provided for the application menu, **Quick Access** toolbar, and ribbon. You can still use 'old style' keyboard shortcuts, such as CTRL + N to open a new file, and CTRL + P to print the current file.

To display keytips, press ALT. The keytips (letters or numbers) are shown on the screen next to the corresponding command or user interface element. Press the displayed accelerator key to immediately invoke the desired command or to show the user interface element. For example, pressing ALT, and then pressing 1 creates a new file.



To hide the keytips, press ALT again.

See also:

[Default Keyboard Shortcuts](#) (page 142)

Navigation Tools

The navigation bar provides access to tools related to interactive navigation and orientation in the model including Autodesk® ViewCube®, SteeringWheels®, and 3Dconnexion® 3D mouse.



You can customize the navigation bar based on what you consider important to show. You can also change the docking position of the navigation bar in the **Scene View**.

The Classic User Interface

If you prefer, you can switch back to the **Classic** user interface, and use the toolbars and pull-down menus from the menu bar instead of the ribbon.

NOTE The **Classic** user interface is no longer being updated with enhancements to Autodesk Navisworks. It is recommended that you work using the standard Autodesk Navisworks interface.

To switch to the classic user interface

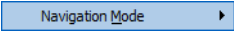
- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **User Interface** option.

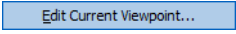
- 3 On the **User Interface** page, select **Classic** from the **User Interface** drop-down list.
- 4 Click **OK**.

Menu Bar

The Menu bar contains all commands available in Autodesk Navisworks, grouped together by similar or 'like' functionality.

For example, all commands related to review functionality are located under the **Review** menu, all commands related to user assistance are located under the **Help** menu and so on.

When a menu has a right-pointing arrow, such as , there is a submenu associated with that choice.

When a menu item is followed by a series of dots, such as , there is a dialog box associated with that choice.

File Menu

This menu contains commands for managing files.

Option	Description
New	Resets the program, and closes the currently open Autodesk Navisworks file.
Refresh	Refreshes your scene with the latest versions of currently loaded model files.
Open	Displays the Open dialog box.
Open URL	Displays the Open URL dialog box.
Append	Displays the Append dialog box.
Merge	Displays the Merge dialog box.

Option	Description
Save	Saves the currently open Autodesk Navisworks file.
Save As	Displays the Save As dialog box.
Publish	Displays the Publish dialog box.
Print	Displays the Print dialog box.
Print Preview	Enables print preview mode.
Print Settings	Displays the Print Setup dialog box.
Delete	Deletes the selected files. You can only delete appended files.
Send by Email	Saves the currently open Autodesk Navisworks file, accesses your email program, and adds the saved file as an email attachment.
Import	Displays the Import dialog box, and enables you to import Intergraph PDS review data, and various Autodesk Navisworks data.
Export	Displays the Export dialog box, and enables you to export data from Autodesk Navisworks.
Recent Files	Displays shortcuts to the most recently opened files.
Exit	Exits the program.

Edit Menu

This menu contains commands for locating, selecting and editing geometry in your model.

Option	Description
Undo	Reverses the last performed operation.
Redo	Reverses the last operation performed by the Undo command.
Select	Gives you access to selection functionality.
Quick Find	Displays the Quick Find dialog box.
Quick Find Next	Repeats the previously run quick find search.
Find Items	Toggles the Find Items control bar.
Find Comments	Toggles the Find Comments dialog box.
Hidden	Toggles hidden mode for selected items.
Required	Toggles required mode for selected items.
Hide Unselected	Toggles hidden mode for unselected items.
Override Item	Enables you to override color, transparency, and transform for selected items.
Reset Item	Enables you to reset selected items back to their original state.
Reset All	Enables you to reset all overridden items back to their original state.

Option	Description
Sort	Enables you to sort the items in the Selection Tree alphabetically.
File Units an Transform	Displays the File Units and Transform dialog box.

View Menu

This menu contains commands that control the Autodesk Navisworks interface.

Option	Description
Control Bars	Enables you to toggle the display of control bars.
Workspaces	Enables you to control workspaces.
Scene View	Enable you to control the views in the Scene View .
Head-Up Display	Enables you to toggle navigation controls, such as ViewCube, Navigation Bar, and HUD elements.
SteeringWheels	Enables you to control the SteeringWheels.
Enable Stereo	Puts the video output into stereo mode.
Stereo Options	Displays the Stereo Options dialog box.
Scene Statistics	Displays useful scene statistics.

Viewpoint Menu

This menu contains a set of commands that affect the current viewpoint, including model appearance, navigation and sectioning.

Option	Description
Saved Viewpoints	Enables you to use saved viewpoints.
Look From	Enables you to look from a preset viewpoint.
Set Viewpoint Up	Sets the viewpoint up vector to align with the selected orientation.
Rendering	Enables you to select rendering mode.
Lighting	Enables you to select lighting mode.
Display	Enables you to display primitives.
Navigation Mode	Enables you to select navigation mode.
Navigation Tools	Enables you to control the camera during interactive navigation.
Sectioning	Enables you to create a limited volume of your model.
Edit Current Viewpoint	Displays the Edit Viewpoint dialog box for the current viewpoint.

Review Menu

This menu contains commands for reviewing and markup.

Option	Description
Comments	Enables you to add and manage comments.

Option	Description
Redline	Enables you to add and manage annotations.
Measure	Enables you to use the measure tools.
Links	Enables you to add and manage links.
Tags	Enables you to locate review tags.
SwitchBack	Sends the current view of the currently loaded file back to AutoCAD or MicroStation-based program.
Renumber Comment IDs	Renumbers all comment IDs, making them unique to the scene.
Renumber Tag IDs	Renumbers all tag IDs, making them unique to the scene.

Tools Menu

This menu contains commands for advanced model analysis and reviewing, and also commands for customizing Autodesk Navisworks.

Option	Description
Clash Detective	Toggles the Clash Detective tool window.
Presenter	Toggles the Presenter tool window.
TimeLiner	Toggles the TimeLiner tool window.
TimeLiner Playback	Toggles the TimeLiner Playback window.
DataTools	Displays the DataTools dialog box.
Animator	Toggles the Animator tool window.

Option	Description
Scripter	Toggles the Scripter tool window.
Batch Utility	Toggles the Batch Utility window.
Compare	Displays the Compare dialog box.
Redline	Toggles the Redline Tools tool window.
Links	Toggles the display of links.
Quick Properties	Toggles the display of quick properties.
Measure	Toggles the Measure Tools tool window.
Animation	Enables you to control animation playback, and record viewpoint animations.
Background	Enables you to select a background color for the Scene View .
File Options	Displays the File Options dialog box.
Customize	Displays the Customize dialog box.
Global Options	Displays the Options Editor .

Toolbars

Autodesk Navisworks toolbars provide quick access to frequently used commands.

Every button on a toolbar includes a tooltip, which describes the function the button activates. Placing the mouse over a button displays a brief instruction on how to use this feature in the **Status** bar.

You can rearrange, open and close toolbars:

- To move a toolbar, click the dotted line at the edge of the toolbar, and drag it to a different location.
- To open or close toolbars, right-click an empty area next to the last toolbar on the screen, and choose from the list of available toolbars on the shortcut menu.

In addition to rearranging the existing Autodesk Navisworks toolbars, you can customize their appearance and content, and create your own toolbars.

NOTE To quickly personalize a toolbar, click the **Toolbar Options**  button on the right, and click **Add or Remove Buttons** on the shortcut menu.

When a Autodesk Navisworks toolbar button has a down-pointing arrow, such as , a submenu toolbar is associated with that choice. Click the triangle to open the menu, and select a specific option. As you move through the menu, additional help is displayed in the **Status** bar. When the option is selected, it becomes the current command and is displayed as a button in the toolbar. To repeat the command, click the button in the toolbar. To choose a different command, click the triangle again.

Some toolbar buttons enable you to choose a program mode. For example, to look around your model, you need to be in look around mode. To rotate the model, you need to be in **Free Orbit** mode and so on. Autodesk Navisworks remains in the selected mode until instructed otherwise. To identify the mode you are in, look at the buttons. If a button is highlighted and has a dark blue boarder around it, the corresponding mode is currently active.



To leave the mode, either click the same button again or choose a different mode.

Some buttons are used to toggle the display of dialog boxes, and dockable windows (for example, the **Presenter** window, the **Animator** window etc.). Again, if a button is highlighted and has a dark blue boarder around it, it means that the corresponding display element is currently open.

As you open more toolbars on the screen, or resize the Autodesk Navisworks window, the toolbars may get overlapped with each other to reduce the screen clutter. When this happens, some buttons will be hidden under the overlaps. To quickly access the entire set of commands on a toolbar, click the chevron

» button at the right end of the toolbar. The remaining commands available for that toolbar will appear.

In this section, you will find a complete list of Autodesk Navisworks toolbars and associated buttons.

NOTE The actual toolbar content can differ from this reference depending on the workspace you use.

Standard Toolbar



This toolbar provides quick access to file management commands. It also enables you to undo/redo your actions, and open the Help system.

Button	Description
	Resets the program, and closes the currently open Autodesk Navisworks file.
	Refreshes your scene with the latest versions of currently loaded model files.
	Displays the Open dialog box.
	Displays the Append dialog box.
	Displays the Merge dialog box.
	Saves the currently open Autodesk Navisworks file.
	Displays the Publish dialog box.
	Saves the currently open Autodesk Navisworks file, accesses your email program,

Button	Description
	and adds the saved file as an email attachment.
	Reverses the last performed operation.
	Reverses the last operation performed by the Undo command.
	Displays the Print dialog box.
	Displays copyright and license information about your copy of Autodesk Navisworks.
	Opens the Help system.

Selection Tools Toolbar



This toolbar provides access to the selection commands, plus enables you to hide geometry objects.

Button	Description
	Turns on Select mode.
	Turns on Select Box mode.
	Toggles required mode for selected items.
	Toggles hidden mode for selected items.
	Toggles hidden mode for unselected items.

Navigation Mode Toolbar



This toolbar includes nine modes and six SteeringWheels for interactive navigation around your 3D models.

Button	Description
	Selects the wheel.
	Turns on Walk mode.
	Turns on Look Around mode.
	Turns on Zoom mode.
	Turns on Zoom Box mode.
	Turns on Pan mode.
	Turns on Orbit mode.
	Turns on Free Orbit mode.
	Turns on Fly mode.
	Turns on Constrained Orbit mode.

Rendering Style Toolbar



This toolbar controls the model appearance in Autodesk Navisworks.

Button	Description
	Selects Lighting mode.
	Selects Rendering mode.
	Toggles the rendering of surfaces.
	Toggles the rendering of lines.
	Toggles the rendering of points.
	Toggles the rendering of snap points.
	Toggles the rendering of 3D text.

Workspace Toolbar



This toolbar gives you quick access to the Autodesk Navisworks review and analysis tools.

Button	Description
	Toggles the Redline Tools tool window.
	Toggles the display of links.
	Toggles the display of quick properties.
	Toggles the Measure Tools tool window.

Button	Description
	Toggles the Viewpoints control bar.
	Toggles the Sectioning toolbar.
	Toggles the Plan View control bar.
	Toggles the Section View control bar.
	Toggles the Selection Tree control bar.
	Toggles the Sets control bar.
	Toggles the Comments control bar.
	Toggles the Find Comments dialog box.
	Toggles the Find Items control bar.
	Toggles the Properties control bar.
	Toggles the Clash Detective tool window.
	Toggles the Presenter tool window.
	Toggles the TimeLiner tool window.
	Toggles the TimeLiner Playback window.
	Toggles the Animator tool window.

Button	Description
	Toggles the Scripter tool window.
	Controls workspaces.

Model Views Toolbar



This toolbar controls the views in the **Scene View**.

Button	Description
	Splits your active scene view vertically.
	Splits your active scene view horizontally.
	Adds title bars to all custom scene views.

Sectioning Toolbar



This toolbar enables you to create a limited volume of your model.

Button	Description
	Toggles sectioning on/off.
	Enables you to link two opposing section planes together.
	Displays the Set Sectioning Box Size dialog box.

Button	Description
	Restricts the section distance to the bounding box limits of the objects selected in the Scene View
	Displays the Sectioning Plane dialog box.
	Enables/disables the current section plane.
	Enables you to choose the alignment method.
	Displays the Section Plane Settings dialog box.

Animation Toolbar



This toolbar allows you to record viewpoint animations, play back object and viewpoint animations, and toggle the scripting functionality.

Button	Description
	Rewinds the current animation back to the beginning.
	Steps back a single animation frame or keyframe.
	Plays the current animation backwards.
	Records the viewpoint animation.
	Pauses the animation.

Button	Description
	Stops playing the current animation, and rewinds it back to the beginning.
	Plays the currently selected animation.
	Steps one frame or keyframe forwards.
	Fast forwards the current animation to the end.
	Toggles the Scripter engine on and off in the Autodesk Navisworks file.

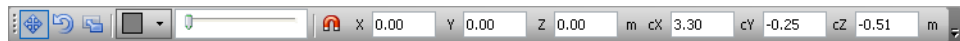
Collaborate Bar Toolbar



This toolbar enables you to participate in design review sessions across a Local Area Network (LAN).

Button	Description
	Starts Windows™ NetMeeting.
	Enables you to take control of the session, and become the 'driver'.
	Refreshes Autodesk Navisworks data on all attendees machines.

Object Manipulation Toolbar



This toolbar enables you to edit geometry in your model.

Button	Description
	Toggles the display of the translation gizmo.
	Toggles the display of the rotation gizmo.
	Toggles the display of the scale gizmo.
	Enables you to apply color override.
	Enables/disables snapping.

Navigation Tools Toolbar



This toolbar enables you to control the camera during interactive navigation.

Button	Description
	Dollies and pans the camera so that the entire model is in the Scene View.
	Zooms the camera so that the selected item fills the Scene View.
	Puts the Scene View into focus mode.
	Holds the selected items. As you move around the model, these objects will move with you.
	Uses a perspective camera.

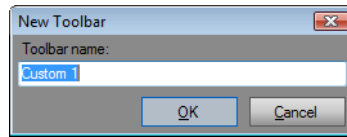
Button	Description
	Uses an orthographic camera.
	Toggles collision.
	Toggles gravity.
	Toggles crouching.
	Toggles third person view.
	Aligns the current viewpoint with the X axis.
	Aligns the current viewpoint with the Y axis.
	Aligns the current viewpoint with the Z axis.

Customize the Toolbars

You can customize the appearance and contents of the Autodesk Navisworks toolbars by using the **Customize** dialog box.

To add a custom toolbar

- 1 Click **Tools ► Customize**.
- 2 In the Customize dialog box, Toolbars tab, click the New button.
- 3 Enter a name for the toolbar in the **Toolbar Name** box. By default, new toolbars are named “Custom X” where “X” is the next available number added to the list.



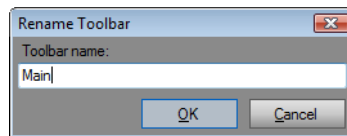
- 4 Click **OK**.
The new toolbar appears in the toolbar area.
- 5 Close the **Customize** dialog box.

To rename a custom toolbar

- 1 Click **Tools ► Customize**.
- 2 In the **Customize** dialog box, **Toolbars** tab, click your toolbar.
- 3 Click **Rename**.

NOTE You can only rename custom toolbars.

- 4 Enter the new name for your toolbar.



- 5 Click **OK**.
- 6 Close the **Customize** dialog box.

To delete a custom toolbar

- 1 Click **Tools ► Customize**.
- 2 In the **Customize** dialog box, **Toolbars** tab, click the toolbar you don't need.
- 3 Click **Delete**.

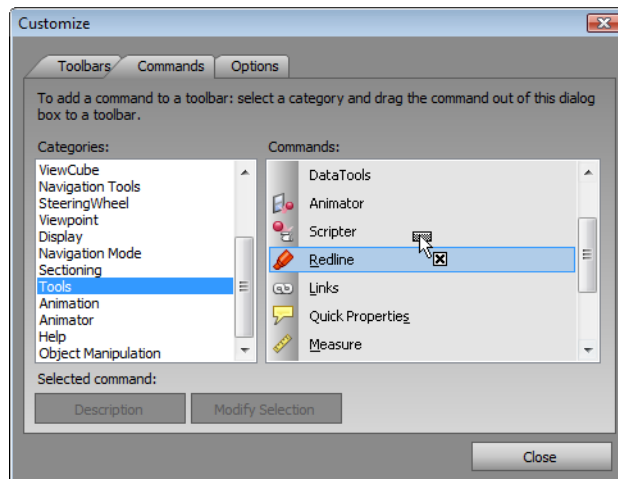
NOTE You can only delete custom toolbars.

- 4 Close the **Customize** dialog box.

To add commands

- 1 Click **Tools ► Customize**. This opens the **Customize** dialog box.

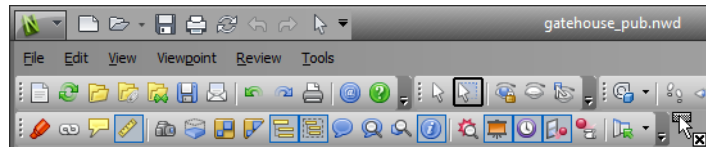
- 2 There are several ways to add commands to a toolbar or a menu:
 - In the **Customize** dialog box, **Commands** tab, click the desired category, and drag the command from the commands list on to the toolbar. If the command has a default icon assigned to it (it is shown next to the command in the command list), the icon will appear as a button on your toolbar. If no icon is assigned to the command, the name of the command will appear as a button on the toolbar. For menus, the command will appear exactly as it is shown in the command list.



- Drag the command from another menu or toolbar onto your toolbar or menu. This moves the command from its original location into a new place.
 - Hold CTRL and drag the command from another menu or toolbar onto your toolbar or menu. This creates a copy of the command, and does not remove the command from its original location.
- 3 If you want to edit the command's appearance, right-click it on the toolbar or menu, and use options on the shortcut menu.
 - 4 Close the **Customize** dialog box.

To delete commands

- 1 Click **Tools ► Customize**. This opens the **Customize** dialog box.
- 2 Drag the command away from the menu or toolbar, until a cursor displays a small cross.

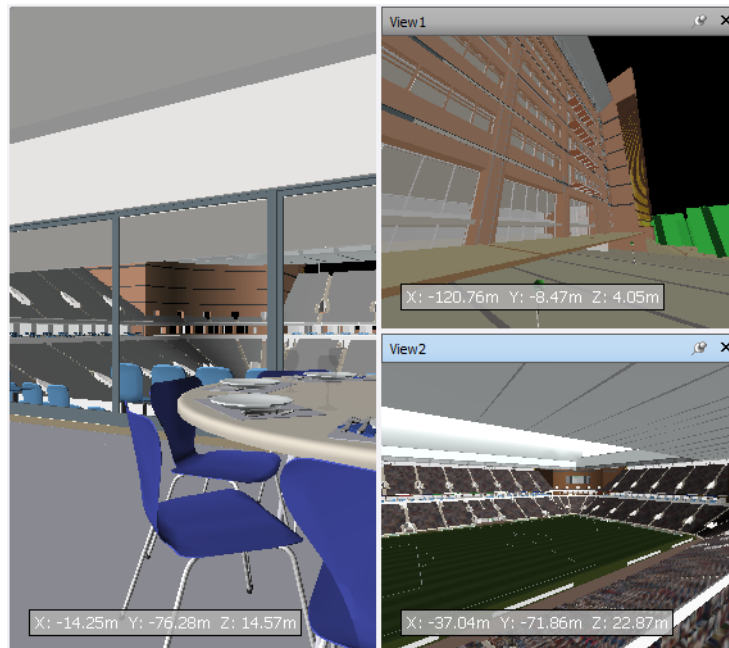


- 3 Release the left mouse button to delete the command.
- 4 Close the **Customize** dialog box.

Scene View

This is the area where you view and interact with your 3D models.

When you start Autodesk Navisworks, the **Scene View** contains only one scene view, but you can add more scene views, if needed. Custom scene views are named "ViewX" where "X" is the next available number. The default scene view cannot be moved.



Looking at several views of your model simultaneously is useful when you compare lighting and rendering styles, animate different parts of your model, and so on.

Only one scene view can be active at a time. A scene view becomes active as you work in it. If you left-click a scene view, the scene view is activated and whatever you click is selected, or, if you click an empty area, everything is deselected. Right-clicking a scene view activates it, and opens a shortcut menu.

Each scene view remembers the navigation mode being used. The recording and playback of animations only occurs in the currently active view.

Each scene view can be resized. To resize scene views, move the cursor over the scene view intersection and drag the splitter bar .

You can make custom scene views dockable. Dockable scene views have title bars, and can be moved, docked, tiled, and auto hidden the same way as [dockable windows](#) (page 132). If you want to use several custom scene views, but don't want to have any splits in the **Scene View**, you can move them elsewhere. For instance, you can tile your scene views on the **Viewpoints** control bar.

NOTE You cannot undock the default scene view.

Full Screen Mode

In **Full Screen** mode your current scene view takes up the full screen.

To interact with the model in the scene view, you can use the ViewCube, the Navigation Bar, the keyboard shortcuts, and the shortcut menu.

TIP If you use two monitors, your default scene view is automatically placed on the primary display, and the interface can be placed on the secondary display to control the interaction.

Scene View Content

Autodesk Navisworks enables you to change the size of content displayed in a scene view. This can be of particular use if you are composing a scene for image or animation export. By setting the content size to the same proportions as your intended output, you can visualize exactly how it will look.

To create a custom scene view

- To split your active scene view horizontally, click **View** tab ► **Scene View** panel ► **Split View** ► **Split Horizontal** .
- To split your active scene view vertically, click **View** tab ► **Scene View** panel ► **Split View** ► **Split Vertical** .

 **Toolbar: Classic** user interface: **Model Views** ► **Split Horizontal** 
and **Model Views** ► **Split Vertical** 

To make a custom scene views dockable

- Click **View** tab ► **Scene View** panel ► **Show Title Bars**.
All of your custom scene views now have title bars.

 **Toolbar: Classic** user interface: **Model Views** ► **Toggle Title Bars** 

To delete a custom scene view

- 1 If your scene view is not dockable, click **View** tab ► **Scene View** panel ► **Show Title Bars**.
- 2 Click  to close the scene view.

NOTE You cannot delete the default scene view.

To toggle Full Screen mode

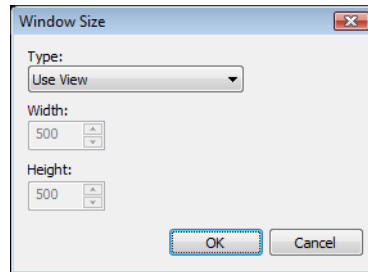
- Click **View** tab ► **Scene View** panel ► **Full Screen** .

 **Command entry:** F11

Shortcut menu: **Viewpoint** ► **Full Screen**

To resize the content of the active scene view

- 1 Click **View** tab ► **Scene View** panel ► **Window Size** .
- 2 In the **Window Size** dialog box, **Type** drop-down list, select the sizing type.



- **Use View** - makes the content fill the currently active scene view.
 - **Explicit** - defines the exact width and height for the content.
 - **Use Aspect Ratio** - uses the aspect ratio of the current scene view to automatically calculate the width of the content when the height is entered, or the height of the content when the width is entered.
- 3 If you selected the **Explicit** option, enter the width and height for your content in pixels.
If you selected the **Use Aspect Ratio**, enter the width or height for your content in pixels.
 - 4 Click **OK**.

Dockable Windows

Most Autodesk Navisworks features are accessible from dockable windows.

There are several windows to choose from, which are grouped into several functional areas:

Main Tools Windows

These windows give you access to the core Autodesk Navisworks functionality:

- **Clash Detective**
- **TimeLiner**
- **Presenter**
- **Animator**
- **Scripter**
- **Appearance Profiler**

Review Windows

These windows contain tools required to perform select/search/review operations:

- **Selection Tree**
- **Sets**
- **Find Items**
- **Properties**
- **Comments**
- **Find Comments**
- **Measure Tools**

Viewpoint Windows

These windows contain tools necessary to set up and use viewpoints:

- **Saved Viewpoints**
- **Tilt** - 3D workspace only.
- **Plan View** - 3D workspace only.
- **Section View** - 3D workspace only.
- **Section Plane Settings** - 3D workspace only

Multi-Sheet Windows

These windows enable you to work with multi-sheet files:

- **Project Browser**
- **Find Item in Other Sheets and Models**

Windows can be moved and resized, and either floated in the **Scene View** or docked (pinned or auto-hidden).

TIP You can quickly dock and undock a window by double-clicking the window's title bar.

A docked window shares one or more edges with adjacent windows and toolbars. If a shared edge is moved, the windows change shape to compensate. You can also float windows anywhere on the screen, if necessary.

NOTE The **Tilt** window can only be docked vertically on the left or right, taking up the full height of the canvas, or be floating.

By default, a docked window is pinned, meaning that the window remains displayed at its current size and can be moved. When you auto hide a window and move the mouse pointer away from it, the window is reduced to a tab displaying the window name. Moving the mouse pointer over the tab displays the window fully, but temporarily, over the canvas. Auto-hiding a window can show more of the canvas while still keeping the window available. Auto-hiding a window also prevents it from being undocked, grouped, or ungrouped.

NOTE When you dock windows inside the default scene view, you do not get pin and auto-hide functionality. This does not affect custom scene views.

An undocked window is one that has been separated from the program window. Each undocked window can be moved around the screen or screens as desired. Although undocked windows cannot be pinned, they can be resized and grouped.

A window group is a way to have more than one window occupy the same amount of space on the screen. When windows are grouped, each window is represented by a tab at the bottom of the group. In a group, click a tab to display that window. You can group or ungroup window as necessary and save custom workspaces. After changing window positions, you can save your settings as a custom workspace.

Auto Hide Position

When you auto hide a window, it collapses against a specific side of the canvas - Top, Left, Right, or Bottom. The side to which it collapses is determined by the docking position. So, for example, if you dock a window to the left of canvas, it collapses to the left.

The Shortcut Menu

Right-clicking a dockable window displays a shortcut menu of available commands. If you right-click a single item, or select one or more items and right-click, this menu contains commands related to the items. If you right-click an area that contains no items or data, the menu contains commands related to the dockable window, if appropriate.

To show a dockable window

- 1 Click **View** tab ► **Workspace** panel ► **Windows** .
- 2 Select the check box next to the desired window in the drop-down list.

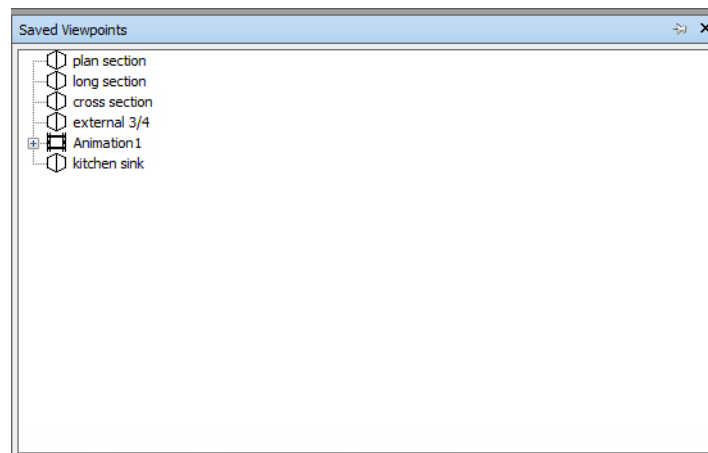
To move a dockable window

- 1 Click and drag the title bar at the top or side of the window.
- 2 Optional: to prevent a window from automatically docking while you drag it, hold down the CTRL key.

TIP The docking tool allows you to place windows in a specific relationship to the canvas areas.

To group dockable windows

- 1 Click and drag the title bar of the window to be added to another window or group.
- 2 Drop the window on the title bar of the receiving window or group. A tab with the name of the dragged window is added to the bottom of the receiving window.



To ungroup dockable windows

- 1 Within the group, click the tab for the window you want to remove.
- 2 Click and drag the window tab out of the group.

- 3 Drop the window to ungroup it.

To auto hide dockable windows

- 1 On a window title bar, click .

The window continues to be displayed until you move the mouse pointer away from it. When you move the mouse pointer, the window is collapsed until you place the mouse pointer over the window tab on the side of the canvas where its docked.

NOTE To move or group windows, you need to pin them first.

To pin dockable windows

- 1 Move the mouse cursor over the title bar to display the hidden window.
- 2 Click  on the title bar. The window is now pinned, and can be moved and grouped.

To resize a dockable window or a group of windows

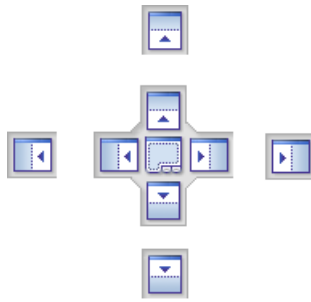
- 1 Place the mouse pointer over a window border until the mouse pointer changes to the splitter bar .
- 2 Click and drag the boarder to the desired size.

TIP You can resize both pinned and auto hidden windows. In an auto-hidden group, each window can be resized separately from other windows. In a pinned group, resizing one window resizes the rest of the windows.

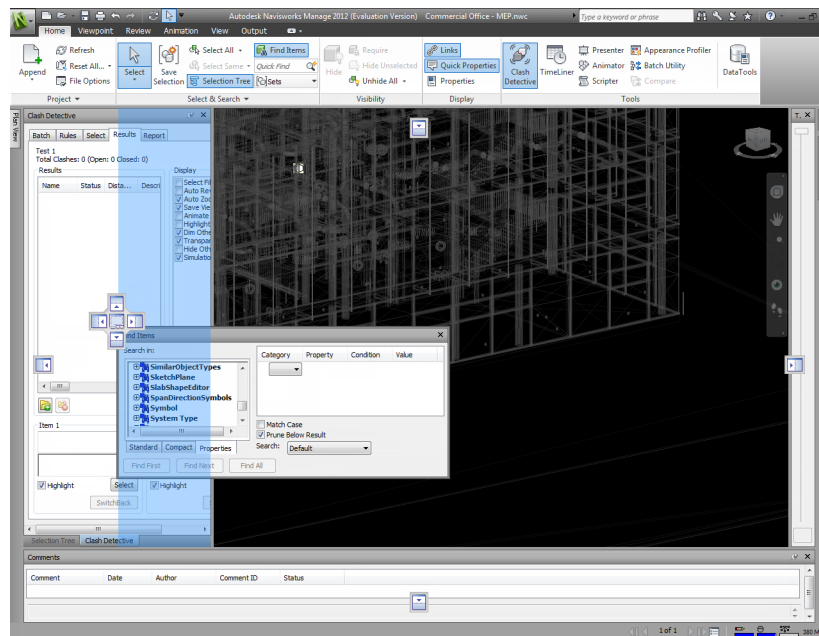
Docking Tool

The docking tool indicates the relationship of a dragged window to the rest of the canvas, and enables you to pinpoint drag and drop destinations.

The tool contains an inner zone and outer zone of controls representing the drop destination. Five stickers of the inner zone are used to dock windows relative to the closest suitable area on the canvas, while four stickers of the outer zone are used to dock windows relative to the canvas itself.



The docking tool also provides visual previews of what space will be occupied by a window. These previews are shown when you are moving a window and while your mouse is over one of the stickers.



TIP To quickly create a window group, use the sticker at the center of the docking tool when you drag a window to its location. This works anywhere on the canvas excluding the default scene view and the **Tilt** window. Custom scene views can be grouped with other windows.

To move a window with the docking tool

- 1 Click and drag the title bar at the top or side of the window towards the place where you want it to dock. This activates the docking tool.
- 2 Drag the window over the sticker on the docking tool that represents the area you want the window to occupy.
- 3 Release the mouse button to dock the window there. The window is automatically resized to fill the area.

Status Bar

The **Status** bar appears at the bottom of the Autodesk Navisworks screen. It cannot be customized or moved around.

The left-hand corner of the **Status** bar is used to display short instructions on how to use the Autodesk Navisworks features (applies to the **Classic** user interface only).

The right-hand corner of the **Status** bar includes four performance indicators that give you constant feedback as to how well Autodesk Navisworks is performing on your machine, a button to toggle the **Project Browser** window, and controls to navigate between sheets/models in multi-sheet files.



Multi-Sheet Navigation Controls

Click the previous/next and first/last arrows to open the desired sheet/model in the **Scene View**. This is equivalent to double-clicking the sheet/model in the **Project Browser** window. The sheet/model will be automatically prepared for use in Autodesk Navisworks, if applicable.

NOTE These controls are only available for multi-sheet files.

Project Browser Button

Click the **Project Browser**  button to toggle the [Project Browser window](#) (page 214).

Pencil Progress Bar

The progress bar under the left hand icon (pencil) indicates how much of the current view is drawn, that is how much drop-out there is in the current

viewpoint. When the progress bar is at 100%, the scene is completely drawn, with no drop-out. The icon changes color when a redraw is in progress. Whilst the scene is being drawn, the pencil will change to yellow. If there is too much data to handle and your machine cannot process this quickly enough for Autodesk Navisworks, then the pencil changes to red, indicating a bottleneck.

Disk Progress Bar

The progress bar under the central icon (disk) indicates how much of the current model is loaded from disk, that is how much is loaded into memory. When the progress bar is at 100%, the entire model, including geometry and property information, is loaded into memory. The icon changes color when a file load is in progress. Whilst data is being read, the disk changes to yellow. If there is too much data to handle and your machine cannot process this quickly enough for Autodesk Navisworks, then the disk changes to red, indicating a bottleneck.

Web Server Progress Bar

The progress bar under the right hand icon (web server) indicates how much of the current model is downloaded, that is how much has been downloaded from a web server. When the progress bar is at 100%, the entire model has been downloaded. The icon changes color when a file load is in progress. Whilst data is being downloaded, the web server changes to yellow. If there is too much data to handle and your machine cannot process this quickly enough for Autodesk Navisworks, then the web server changes to red, indicating a bottleneck.

Memory Bar

The field to the right of the icons reports the amount of memory currently being used by Autodesk Navisworks. This is reported in Megabytes (MB).

Undo/Redo Commands

You can undo or redo your actions in Autodesk Navisworks.

The default settings are adequate for regular Autodesk Navisworks usage, but you can [adjust](#) (page 765) the amount of space allocated to the undo/redo buffer, if necessary.

To undo an action

- Click **Undo**  on the **Quick Access** toolbar.

 **Command entry:** CTRL + Z

 **Toolbar:** **Classic** user interface: **Standard** > **Undo** 

To redo an action

- Click **Redo**  on the **Quick Access** toolbar.

 **Command entry:** CTRL + Y

 **Toolbar:** **Classic** user interface: **Standard** > **Redo** 

Autodesk Navisworks Workspaces

Workspaces retain information about which windows are open, their positions, and the size of the application window.

Workspaces retain changes made to the ribbon but not to the **Quick Access** toolbar.

NOTE In the **Classic** user interface mode (that is, the ribbon is turned off), workspaces retain information about the dockable windows and the toolbars.

The workspaces can be shared with other users. You could, for example, create separate workspaces for occasional and power Autodesk Navisworks users, or setup your own corporate standard.

Autodesk Navisworks comes with several pre-configured workspaces:

- **Safe Mode** - selects the layout with the minimum features.
- **Navisworks Extended** - selects the layout recommended for advanced users.
- **Navisworks Standard** - selects the layout with commonly-used windows auto-hidden as tabs.
- **Navisworks Minimal** - selects the layout giving the most space to the **Scene View**.

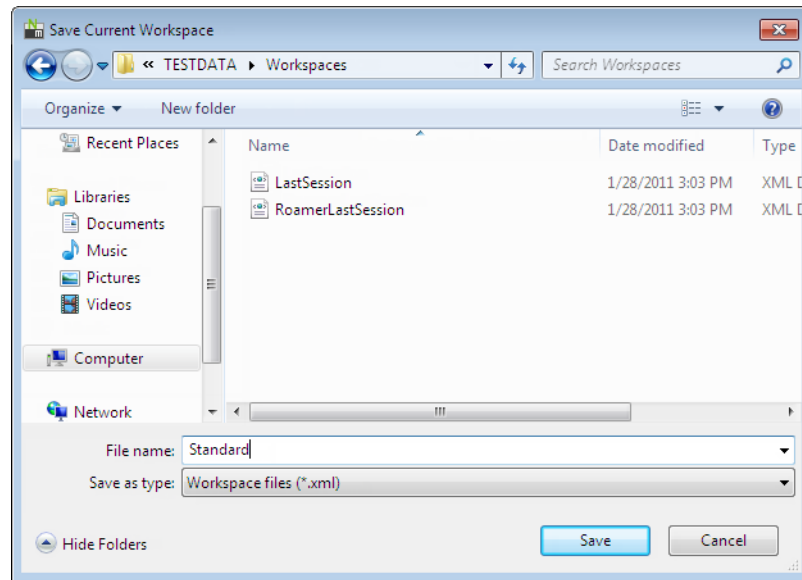
You can use these workspaces as-is or modify them in accordance to your requirements. When you first start Autodesk Navisworks, the **Navisworks Minimal** workspace is used. You can choose a different workspace at any

time by clicking **View** tab ► **Workspace** panel ► **Load Workspace** , and then selecting the required workspace from the list.

 **Toolbar: Classic** user interface: **Workspace** ► **Workspaces** 

To save current layout to a new workspace

- 1 Set up your design review layout. For example, you can group together the Properties and Saved Viewpoints windows.
If you use the **Classic** user interface mode, for example, you can close all toolbars except the **Standard**, **Selection Tools**, **Navigation Mode**, and **Workspace**.
- 2 Click **View** tab ► **Workspace** panel ► **Save Workspace** .
- 3 In the **Save Current Workspace** dialog box, enter a name for the new workspace. You can also select the name of an existing workspace to overwrite it with your modified configuration.

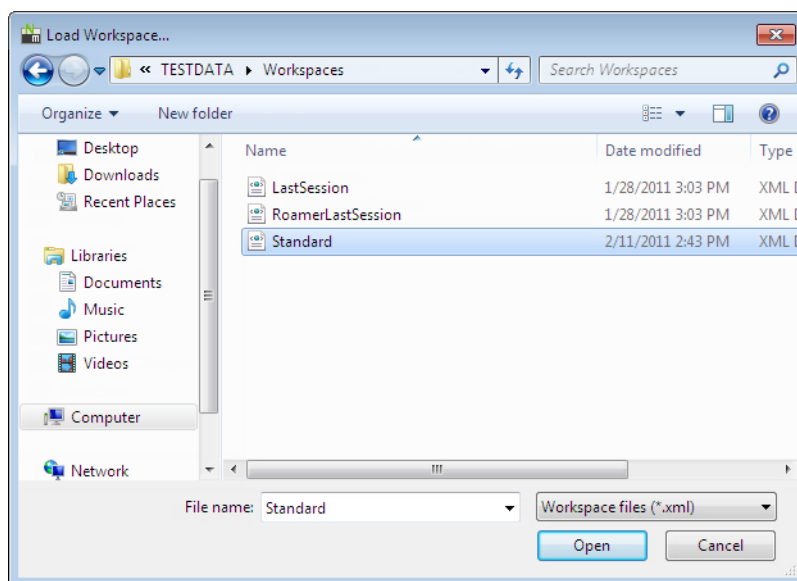


- 4 Click **Save**.

To load a saved workspace into Autodesk Navisworks

- 1 Click **View** tab ► **Workspaces** panel ► **Load Workspace** .

- 2 In the **Load Workspace** dialog box, browse to the folder containing the desired workspace, and select it.



- 3 Click **Open**.

Default Keyboard Shortcuts

Keyboard shortcuts are keyboard alternatives you can use to initiate commands normally accessed with the mouse.

For example, to open the **Selection Tree** window you can press CTRL + F12, to open the **Comments** window, you can press SHIFT + F6, and so on. Keyboard shortcuts offer a means to let you work faster and more efficiently. Some dialog boxes or dockable windows can be closed with the same command used to open it.

Many keyboard shortcuts are already set for most commonly used actions. You can modify the default shortcuts or add new shortcuts, if necessary.

Default Keyboard Shortcut	Description
PgUp	Zooms to view all objects in the Scene View.

Default Keyboard Shortcut	Description
PgDn	Zooms to magnify all selected objects in the Scene View.
HOME	Takes you to Home view. This keyboard shortcut only applies to the Scene View windows. This means it will only work when this window has focus.
ESC	Deselects everything.
SHIFT	Used to modify the middle mouse button actions.
CTRL	Used to modify the middle mouse button actions.
ALT	Turns the keytips on or off.
ALT + F4	Closes the currently active dockable window when it is undocked, or exits the application if the main application window is active.
CTRL + 0	Turns on Turntable mode.
CTRL + 1	Turns on Select mode.
CTRL + 2	Turns on Walk mode.
CTRL + 3	Turns on Look Around mode.
CTRL + 4	Turns on Zoom mode.
CTRL + 5	Turns on Zoom Window mode.
CTRL + 6	Turns on Pan mode.
CTRL + 7	Turns on Orbit mode.
CTRL + 8	Turns on Free Orbit mode.
CTRL + 9	Turns on Fly mode.

Default Keyboard Shortcut	Description
CTRL + A	Displays the Append dialog box.
CTRL + D	Toggles Collision mode. You must be in appropriate navigation mode (that is, Walk or Fly) for this keyboard shortcut to work.
CTRL + F	Displays the Quick Find dialog box.
CTRL + G	Toggles Gravity mode.
CTRL + H	Toggles Hidden mode for selected items.
CTRL + I	Displays the Insert File dialog box.
CTRL + M	Displays the Merge dialog box.
CTRL + N	Resets the program, closes the currently open Autodesk Navisworks file, and creates a new file.
CTRL + O	Displays the Open dialog box.
CTRL + P	Displays the Print dialog box.
CTRL + R	Toggles Require mode for selected items.
CTRL + S	Saves the currently open Autodesk Navisworks file.
CTRL + T	Toggles Third Person mode.
CTRL + Y	Reverses the last operation performed by the Undo command.
CTRL + Z	Reverses the last performed operation.
CTRL + PgUp	Displays the previous sheet.
CTRL + PgDn	Displays the next sheet.

Default Keyboard Shortcut	Description
CTRL + F1	Opens the Help system.
CTRL + F2	Opens the Clash Detective window.
CTRL + F3	Toggles the TimeLiner window.
CTRL + F4	Toggles the Presenter window.
CTRL + F5	Toggles the Animator window.
CTRL + F6	Toggles the Scripter window.
CTRL + F7	Toggles the Tilt window.
CTRL + F8	Applies to the Classic user interface. Toggles the Sectioning toolbar.
CTRL + F9	Toggles the Plan View window.
CTRL + F10	Toggles the Section View window.
CTRL + F11	Toggles the Saved Viewpoints window.
CTRL + F12	Toggles the Selection Tree window.
CTRL + HOME	Dollies and pans the camera so that the entire model is in view.
CTRL + Right Arrow	Play selected animation.
CTRL + Left Arrow	Reverse Play selected animation.
CTRL + Up Arrow	Record viewpoint animation.
CTRL + Down Arrow	Stop playing animation.

Default Keyboard Shortcut	Description
CTRL + Space	Pause playing animation.
CTRL + SHIFT + A	Opens the Animation Export dialog box.
CTRL + SHIFT + C	Opens the Export dialog box and enables you to export current search.
CTRL + SHIFT + I	Opens the Image Export dialog box.
CTRL + SHIFT + R	Opens the Export Rendered Image dialog box.
CTRL + SHIFT + S	Opens the Export dialog box and enables you to export search sets.
CTRL + SHIFT + T	Opens the Export dialog box and enables you to export the current TimeLiner schedule.
CTRL + SHIFT + V	Opens the Export dialog box and enables you to export viewpoints.
CTRL + SHIFT + W	Opens the Export dialog box and enables you to export viewpoint report.
CTRL + SHIFT + HOME	Sets current view as Home.
CTRL + SHIFT + END	Sets current view as Front.
CTRL + SHIFT + Left Arrow	Takes you to the previous redline tag.
CTRL + SHIFT + Right Arrow	Takes you to the next redline tag.
CTRL + SHIFT + Up Arrow	Takes you to the first redline tag.
CTRL + SHIFT + Down Arrow	Takes you to the last redline tag.
F1	Opens the Help system.
F2	Renames the selected item, when appropriate.

Default Keyboard Shortcut	Description
F3	Repeats the previously run Quick Find search.
F5	Refreshes your scene with the latest versions of currently loaded model files.
F11	Toggles Full Screen mode.
F12	Opens the Options Editor .
SHIFT + W	Opens the last used SteeringWheel .
SHIFT + F1	Enables you to get context-sensitive help.
SHIFT + F2	Toggles the Sets window.
SHIFT + F3	Toggles the Find Items window.
SHIFT + F4	Toggles the Find Comments window.
SHIFT + F6	Toggles the Comments window.
SHIFT + F7	Toggles the Properties window.
SHIFT + F10	Opens a shortcut menu.
SHIFT + F11	Opens the File Options dialog box.

Navigation with the Wheel Button

If you have a wheel mouse, you can use the middle mouse button to zoom, pan, and orbit.

To	Do this...
Zoom in	scroll the wheel button forward.

To	Do this...
Zoom out	scroll the mouse wheel backward.
Pan	hold down the middle mouse button, and then move the mouse to pan.
Orbit	press and hold SHIFT and hold down the middle mouse button, and then move the mouse to orbit about the currently defined pivot point. This function is not available in a 2D workspace.
Change the pivot point	press and hold the SHIFT and CTRL keys and hold down the middle mouse button, then drag to the point on the model you want to use as the pivot point. This function is not available in a 2D workspace.

NOTE The above does not apply when using Walk, Fly, or any of the classic navigation modes, all of which have their own wheel/middle button behaviors.

Mouse Wheel / Middle Button Navigation

NOTE In a 2D workspace you can only pan and zoom. Also, the Shift+ Middle Button function is only available in a 3D workspace.

	Wheel	Middle But- ton	SHIFT + Middle Button

Non-Navigational Tools

Select	Zoom In/Out	Pan	Orbit
Measure	Zoom In/Out	Pan	Orbit
Redline	Zoom In/Out	Pan	Orbit

Classic Navigation Modes

	Wheel	Middle Button	SHIFT + Middle Button
Walk	Look Up/Down	Glide Camera	Glide Camera (Faster)
Look Around	Zoom In/Out	Pan	Orbit
Zoom	Zoom (Roll)	Zoom (Drag)	-
Zoom Box	Zoom (Roll)	Zoom (Drag)	-
Pan	Zoom (Roll)	Zoom (Drag)	-
Orbit	Zoom (Roll)	Glide Camera	-
Examine	Zoom (Roll)	Pan	-
Fly	-	Roll	-
Turntable	Tilt Up/Down	Pan	-

Standard Navigation Modes

Pan	Zoom In/Out	Pan	Orbit
Zoom Window	Zoom In/Out	Pan	Orbit
Zoom	Zoom In/Out	Pan	Orbit
Orbit	Zoom In/Out	Pan	Orbit
Free Orbit	Zoom In/Out	Pan	Orbit
Constrained Orbit	Zoom In/Out	Pan	Orbit
Look At	Zoom In/Out	Pan	Orbit

	Wheel	Middle Button	SHIFT + Middle Button
SteeringWheels	Zoom In/Out	Pan	Orbit

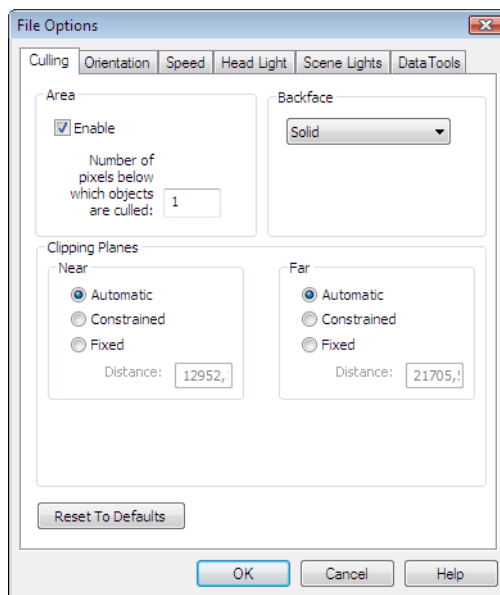
Autodesk Navisworks Options

There are two types of options: **File Options** and **Global Options**.

File Options

For each Autodesk Navisworks file (NWF and NWD), you can adjust the model appearance and the speed of navigation around it. Viewing options are stored with Autodesk Navisworks files (NWF or NWD), and reloaded each time you open these files.

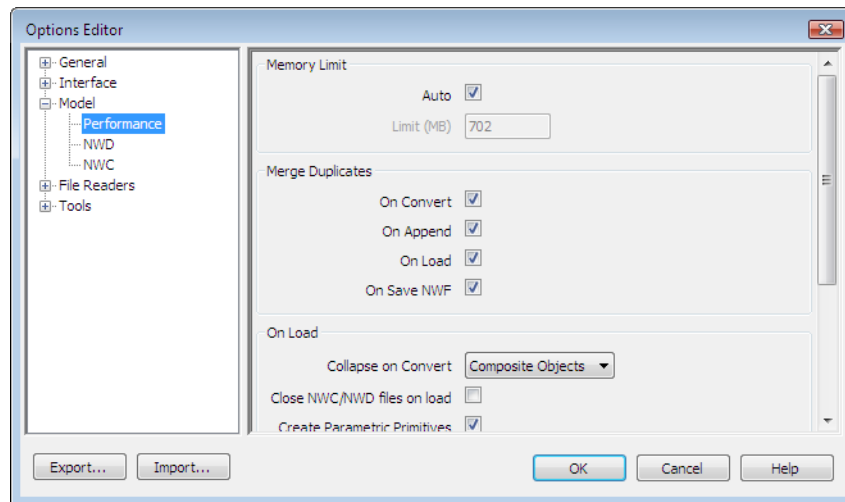
The **File Options** dialog box is used to customize various file options, and can be accessed by clicking **Home** tab ► **Project** panel ► **File Options** .



Global Options

Global options, on the other hand, are set for all Autodesk Navisworks sessions.

The **Options Editor** can be accessed by clicking the application button  **► Options**, or it can be launched as a separate application. To do this, click **Start ► All Programs ► Autodesk ► Navisworks Manage 2012 ► Options Editor**. The options are grouped together, and presented in a tree structure, making it quicker to find and change them.



Global options can be exported and imported, making it quick and easy for project managers, or systems administrators, to ensure the Autodesk Navisworks settings on all machines are identical.

To configure file options

- 1 Click **Home** tab **► Project** panel **► File Options** .
- 2 Use the **File Options** dialog box to customize various file settings.
- 3 Click **OK** to save the changes.

 **Menu:** Classic user interface: **Tools ► File Options**

See also:

[File Options Dialog Box](#) (page 754)

To configure global options

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the desired node, and click the option you want to configure.
- 3 Click **OK** to save the changes.

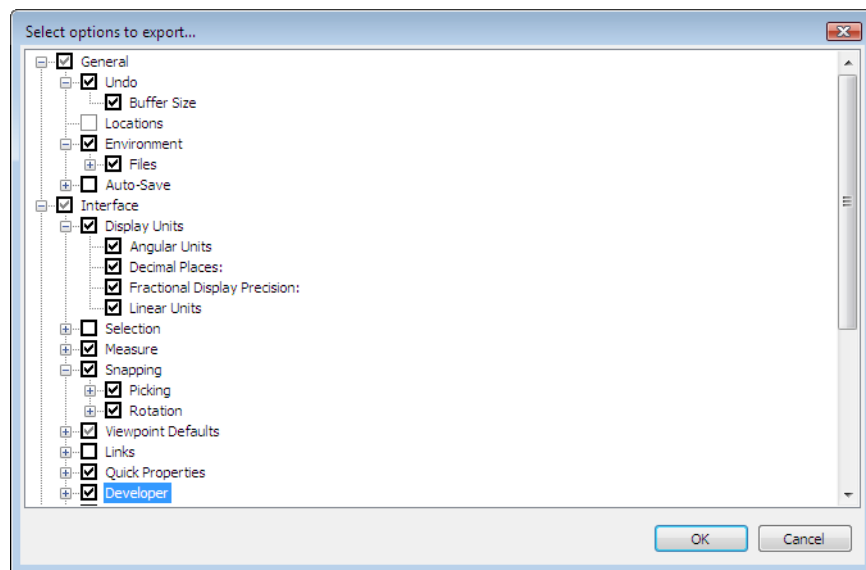
 **Menu:** **Classic** user interface: **Tools** ► **Global Options**

See also:

[Options Editor Dialog Box](#) (page 764)

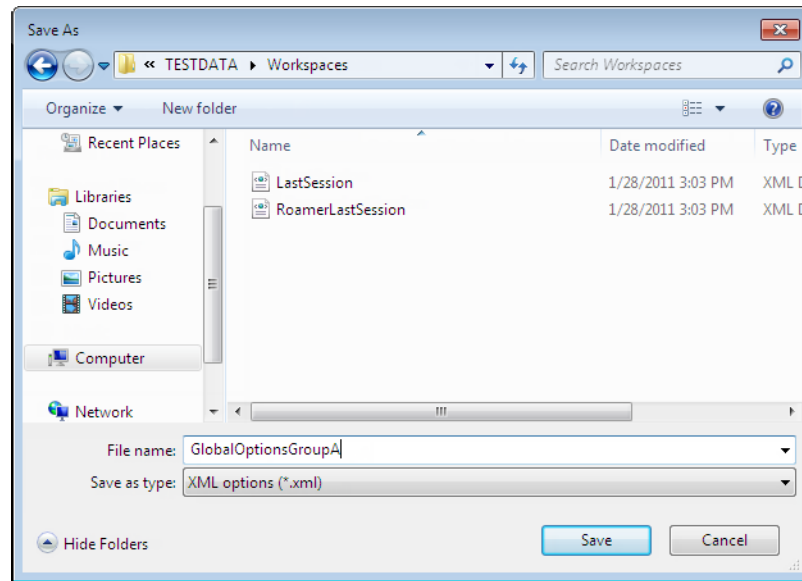
To export global options

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, click the **Export** button.
- 3 In the **Select Options to Export** dialog box, select the check boxes for all options you want to be exported (or “serialized”). If an option cannot be exported, it is greyed out.



TIP To quickly select/deselect all options for a given category, use the top-level check boxes. For example, selecting the **General** check box, instantly selects all options under this node.

- 4 Click **OK** to export the selected settings.
- 5 In the **Save As** dialog box, enter a name for the settings file. You can also select the name of an existing settings file to overwrite it with your modified configuration.

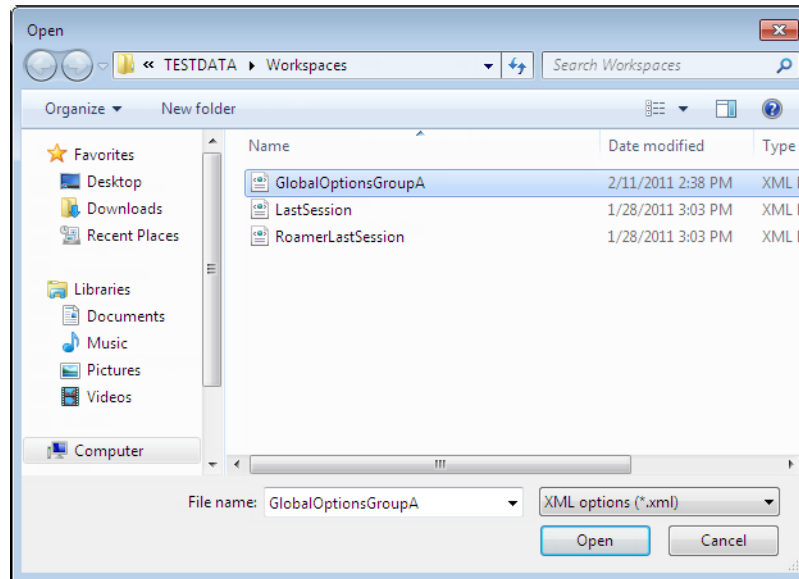


- 6 Click **Save**.
- 7 Click **OK** to close the **Options Editor**.

Menu: Classic user interface: Tools > Global Options

To import global options

- 1 Click the application button  > **Options**.
- 2 In the **Options Editor**, click the **Import** button.
- 3 In the **Open** dialog box, browse to the folder containing the settings file, select it, and click **Open**.



4 Click **OK** to close the **Options Editor**.

 **Menu: Classic user interface: Tools > Global Options**

Location Options

These options enable centralized sharing of global Autodesk Navisworks settings, workspaces, datatools, avatars, Clash Detective rules, Presenter archives, custom Clash Detective tests, object animation scripts, and so on, with other users.

The settings can be shared across an entire project site, or across a specific project group depending on the required level of granularity.

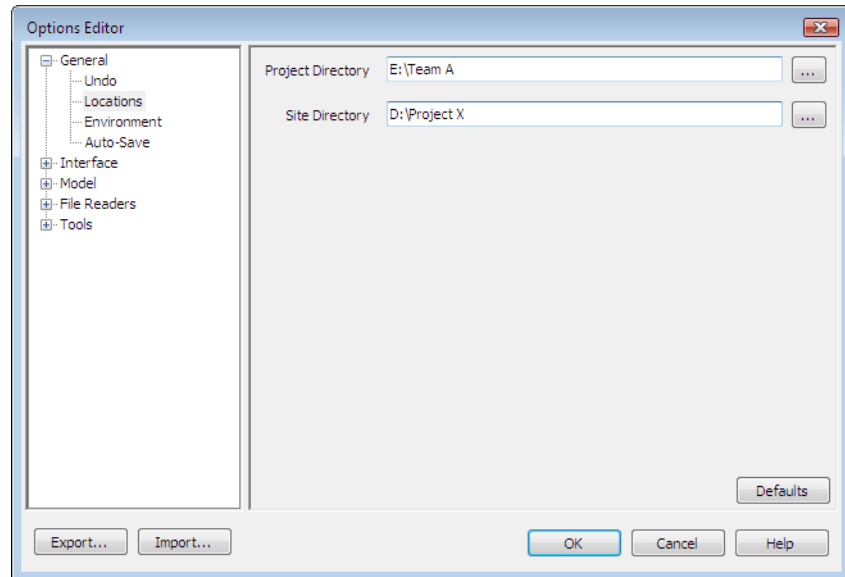
See also:

[How do I share the Autodesk Navisworks settings on a site and project basis?](#) (page 73)

To configure location options

1 Click the application button  **> Options**.

- 2 Expand the **General** node in the **Options Editor**, and click the **Locations** option.



- 3 In the **Project Directory** box, browse to the directory that contains the Autodesk Navisworks settings specific to your project group.
- 4 In the **Site Directory** box, browse to the directory that contains the Autodesk Navisworks settings standard across the entire project site.
- 5 Click **OK**.

 Menu: Classic user interface: Tools ► Global Options

NOTE When you run Autodesk Navisworks for the first time, the settings are picked up from the installation directory. Subsequently, Autodesk Navisworks examines the current user profile and the all users profile on the local machine, and then checks the settings in the **Project Directory** and the **Site Directory**. The files in the **Project Directory** take precedence.

Graphics System

Autodesk Navisworks Manage 2012 supports two graphics systems: **Presenter** Graphics and Autodesk Graphics.

By default, Autodesk Navisworks automatically chooses the most appropriate graphics system to use. For example, 3D models are rendered with **Presenter** graphics, with the exception of 3D FBX files with consistent materials, which will use Autodesk graphics. 2D sheets are rendered with Autodesk graphics. You can use **Options Editor** to specify which graphics system is used.

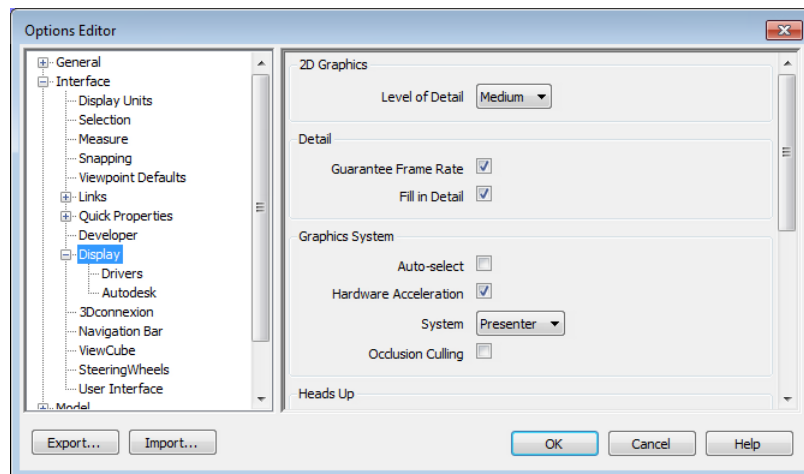
Presenter materials are only available when using Presenter graphics system, and Autodesk materials (also referred to as consistent materials) are only available when using Autodesk graphics system. Real-time navigation is supported by both graphics systems; but you need to use Presenter graphics system if you want to create photorealistic visualizations.

Supported Drivers

You can see a list of all supported drivers in the **Options Editor** (**Interface > Display > Drivers**). By default, all driver options are selected.

To specify graphics system

- 1 Click the application button  > **Options**.
- 2 Expand the **Interface** node in the **Options Editor**, and click the **Display** option.



- 3 In the **Graphics System** area, clear the **Auto-Select** check box. The **System** drop-down box is now active.

- 4 Use the System drop-down box to choose the graphics system you want Autodesk Navisworks to use.
- 5 Click OK.

Display Units

Display units determine the scale of your model in Autodesk Navisworks.

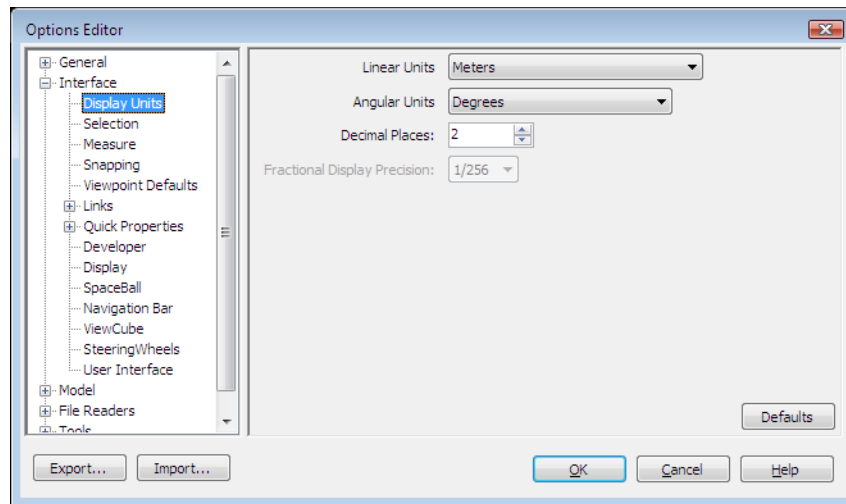
Display units are used to measure geometry in your scene, align appended models, set tolerances for clash detection, set texture sizes and so on.

When you open CAD and laser scan files, Autodesk Navisworks reads the units directly from the files. If this is not possible (for example, the file is unitless), Autodesk Navisworks uses the default units configured for that file type in the **Options Editor** whenever possible. Loaded files are scaled appropriately to the configured display units.

It is possible to rescale the file units, if they are found to be incorrect for the scene.

To customize display units

- 1 Click the application button  ► **Options**.
- 2 Expand the **Interface** node in the **Options Editor**, and click the **Display Units** option.



- 3 Select the **Linear Units** from the drop-down list. Be sure to choose the exact format required.
- 4 Select the **Angular Units** from the drop-down list.
- 5 Enter the number of decimal places you want to see throughout the interface for your units in the **Decimal Places** box. If the unit chosen is a fractional unit, rather than a decimal unit, then you have the choice of what level of fraction to display the units from the **Fractional Display Precision** drop-down list.
- 6 Click **OK**.

 **Menu:** Classic user interface: **Tools** ► **Global Options**

Profiles

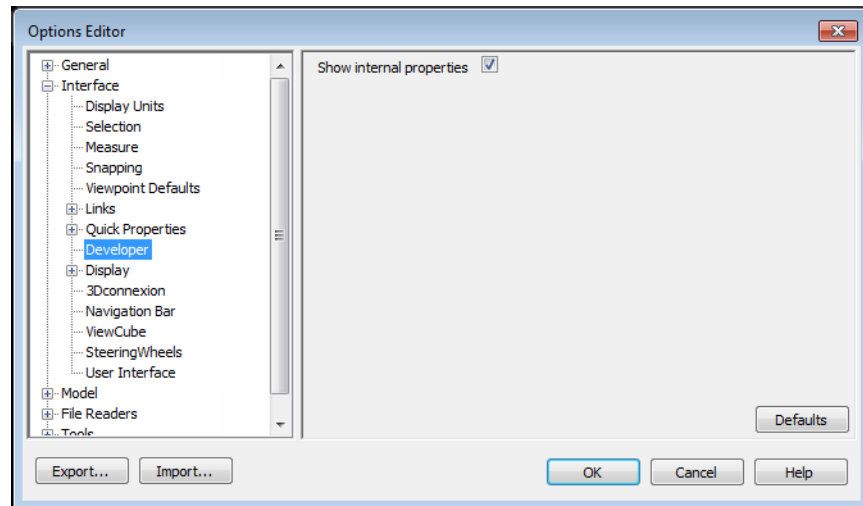
Autodesk Navisworks can be adjusted to your level of CAD technical knowledge.

By default, a standard profile is used. If necessary, you can use a developer profile to display additional object properties.

To use a developer profile

- 1 Click the application button  ► **Options**.

- 2 Expand the **Interface** node, and click the **Developer** option.
- 3 Select the **Show Internal Properties** check box to add additional object properties to the **Properties** window.



- 4 Click **OK**.

 **Menu: Classic user interface: Tools ► Global Options**

Search Directories

Autodesk Navisworks searches for a variety of configuration files in subdirectories of three standard directories.

These files can be overridden on a per user, all users or per installation basis. The search directories are:

- *Application Data\Autodesk Navisworks Manage 2012* within the current user profile. For example, *C:\Documents and Settings\user\Application Data\Autodesk Navisworks Manage 2012* where **user** is the name of the current user.
- *Application Data\Autodesk Navisworks Manage 2012* within the all users default profile. For example, *C:\Documents and Settings\All Users\Application Data\Autodesk Navisworks Manage 2012*.
- Within the Autodesk Navisworks install directory. For example, *C:\Program Files\Autodesk Navisworks Manage 2012*.

NOTE If you are using Windows 7 then the search directory paths are different. Instead of `\Users\Application Data\Autodesk Navisworks Manage 2012`, the path will be `\Users\AppData\Roaming\Autodesk Navisworks Manage 2012`.

Two additional directories, **Site** and **Project**, may be used to share various configuration settings with other users. When you run Autodesk Navisworks for the first time, the settings are picked up from the installation directory. Subsequently, Autodesk Navisworks examines the current user profile and the all users profile on the local machine, and then checks the settings in the Project Directory and the Site Directory. The files in the Project Directory take precedence.

See also:

[Location Options](#) (page 154)

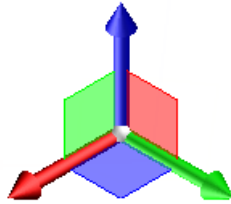
Gizmos

Autodesk Navisworks provides you with gizmo-based tools to interact with 3D objects. The following types of gizmos are used:

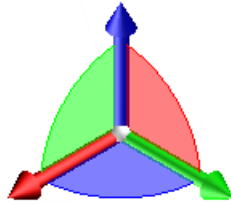
- Transform gizmos. Manipulate objects' transforms (translation, rotation, and scale) globally (as if they'd been changed in the original CAD model). See [Transform Objects](#) (page 383).
- Animation gizmos. Manipulate objects' transforms temporarily for animation purposes. See [Manipulate Geometry Objects](#) (page 531).
- Sectioning gizmos. Manipulate section planes and section box. See [Move and Rotate Section Planes](#) (page 461) and [Enable and Use Section Box](#) (page 465).

Each gizmo displays three colored axes at the correct angles relevant to the current camera position. Gizmos act like 3D objects in that the axis rotate with the viewpoint. However, they are overlaid over the top of the 3D scene,

and can't be obscured by other objects. When you mouse over a grabable part of the gizmo, the cursor changes to a hand  icon.



Move Gizmo



Rotate Gizmo



Scale Gizmo

When you use gizmos, you can adjust snapping to control the precision of your operations (click the application button  **► Options ► Interface** node **► Snapping** page).

Work with Files

4

You can combine these files together, and create a single Autodesk Navisworks file with a whole-project view of your model. This file brings together geometry and data created by multi-disciplinary teams, and enables you to explore and review complex models in real-time.

Native File Formats

Autodesk Navisworks has three native file formats: NWD, NWF, and NWC.

NWD File Format

An NWD file contains all model geometry together with Autodesk Navisworks-specific data, such as review markups. You can think of an NWD file as a snapshot of the current state of the model.

NWD files are very small, as they compress the CAD data by up to 80% of the original size.

NWF File Format

An NWF file contains links to the original native files (as listed on the **Selection Tree**) together with Autodesk Navisworks-specific data, such as review markups. No model geometry is saved with this file format; this makes an NWF considerably smaller in size than an NWD.

NWC File Format (Cache Files)

By default, when you open or append any native CAD or laser scan files in Autodesk Navisworks, a cache file is created in the same directory and with the same name as the original file, but with an .nwc file extension.

NWC files are smaller than the original files, and speed up your access to commonly used files. When you next open file or append file in Autodesk Navisworks, the data is read from the corresponding cache file if it is newer than the original file. If the cache file is older, which means the original file has changed, Autodesk Navisworks converts the updated file, and creates a new cache file for it.

See also:

[NWC File Options](#) (page 790)

[NWD File Options](#) (page 790)

Compatible CAD Applications

The table below is provided as a quick reference only, and does not contain an exhaustive list of compatible CAD applications. For additional information, please refer to the Features section on our website <http://www.autodesk.com/navisworks>.

Application	File Formats
Autodesk AutoCAD	DWG, DXF, 3DS
Autodesk ADT/AutoCAD Architecture	DWG, DXF, 3DS
Autodesk Building Systems/AutoCAD MEP	DWG, DXF, 3DS
Autodesk Inventor	IPT, IAM, IPJ, IGES, STEP
Autodesk AutoCAD Civil 3D	DWG, DXF, 3DS
Autodesk MDT	DWG, DXF, 3DS, VRML
Autodesk Revit	DWG, DXF, DGN
Autodesk Revit Building/Revit Architecture	DWG, DXF, DGN
Autodesk Revit Structure	DWG, DXF, DGN, CIS/2

Application	File Formats
Autodesk Revit Systems/Revit MEP	DWG, DXF, DGN
Autodesk 3DS MAX	DWG, DXF, 3DS, IGES
Autodesk VIZ	DWG, DXF, 3DS
Autodesk Maya	DXF, IGES, VRML
Bentley AutoPLANT	DGN, DWG, DXF
Bentley MicroStation SE	DGN, DWG, DXF, IGES, VRML
Bentley MicroStation J	DGN, DWG, DXF, IGES, STEP, VRML
Bentley MicroStation	DGN, DWG, DXF, IGES, STEP, VRML
Bentley Triforma J	DGN, DWG, DXF, IGES, STEP, VRML
Bentley Triforma	DGN, DWG, DXF, IGES, STEP, VRML
Graphisoft ArchiCAD	DWG, DXF, DGN, IFC
Graphisoft Constructor	DWG, DXF, DGN
AceCad StruCad	DXF
auto.des.sys form.z	DWG, DXF, 3DS, IGES, STL, VRML
AEC CADPIPE	DWG, DXF, 3DS
AVEVA PDMS	RVM
CADopia IntelliCAD	DWG, DXF

Application	File Formats
CEA Technology Plant-4D	DWG, DGN
COADE CADWorx Plant	DWG, DXF, 3DS
COADE CADWorx Pipe	DWG, DXF, 3DS
COADE CADWorx Steel	DWG, DXF, 3DS
COINS BSLink	DWG, DXF, 3DS
COINS Framing	DWG, DXF, 3DS
CSC 3D+	DWG
Dassault Systemes CATIA	DXF, IGES, STEP
Google SketchUp	SKP, DWG, 3DS
Hannappel Software GmbH elcoCAD R4	DWG, DXF, 3DS
Integrapp PDS	DRI, DRV, DWG, DXF, DGN, IGES, STEP
Informatix MicroGDS	MAN
ITandFactory CADISON	DWG, DXF, 3DS
ITandFactory TRICAD MS	VRML
Kiwi Software GmbH ProSteel 3D	DWG, DXF, 3DS
Kubotek USA CADKEY	DWG, DXF, IGES, STEP
M.A.P. CAD-Duct	DWG, DXF, 3DS

Application	File Formats
McNeel North America Rhino	DWG, DXF, 3DS, IGES, STEP
Mensch und Maschine RoCAD	DWG, DXF, 3DS
MultiSUITE MultiSTEEL	DWG, DXF, 3DS
Nemetschek Allplan	DWG, DXF, DGN, IFC
PROCAD 3DSMART	DWG, DXF, 3DS
PTC Pro/ENGINEER	IGES, STEP, VRML
PTC CADD5 5	IGES, STEP
QuickPen PipeDesigner 3D	DWG, DXF, 3DS
QuickPen DuctDesigner 3D	DWG, DXF, 3DS
RAM International CADstudio	DWG, DXF, 3DS
SolidWorks	DWG, DXF, IGES, STEP, VRML
Tekla Structures	DGN, VRML, CIS/2
Tekla Xsteel	DGN
think3 thinkdesign	DWG, DXF, IGES, STEP
UGS I-deas	DXF, IGES, STEP
UGS Solid Edge	IGES, STEP
NX	DXF, IGES, STEP

Application	File Formats
UGS Factory CAD	DWG, DXF, 3DS
UHP Process Piping FabPro Pipe	DWG, DXF, 3DS
UHP Process Piping x-plant	DWG, DXF, 3DS

Supported CAD File Formats

See the following table for the native CAD file formats you can open in Autodesk Navisworks without having the CAD applications installed on your machine.

Format	Extension
Autodesk Navisworks	.nwd, .nwf, .nwc
Autodesk	.fbx
AutoCAD	.dwg, .dxf
ACIS SAT	.sat
CIS/2	.stp, .step
DWF	.dwf
IFC	.ifc
IGES	.igs, .iges
Informatix MicroGDS	.man, .cv7
Inventor	.ipt, .iam, .ipj

Format	Extension
JTOpen	.jt
MicroStation (SE, J, V8, XM)	.dgn, .prp, .prw
Parasolid	.x_b
PDS Design Review	.dri
RVM	.rvm
SketchUp	.skp
STEP	.stp, .step
STL	.stl
VRML	.wrl, .wrz
3D Studio	.3ds, .prjv

Supported Laser Scan File Formats

See the following table for the laser scan file formats you can open in Autodesk Navisworks.

Format	Extension
ASCII Laser File	.asc, .txt
Faro	.fls, .fws, .iQscan, .iQmod, .iQwsp
Leica	.pts, .ptx
Riegl	.3dd

Format	Extension
Z+F	.zfc, .zfs

NOTE Native Trimble file format is not supported. To open your file in Autodesk Navisworks, convert it into ASCII laser file format first.

Use File Readers

Autodesk Navisworks provides file readers to support a variety of CAD file formats and laser scan file formats.

When you open a CAD file in Autodesk Navisworks, an appropriate file reader is automatically used. If necessary, you can adjust the default file reader settings to improve the conversion quality.

3DS File Reader

3DS is a common file format supported by many CAD applications.

The Autodesk Navisworks file reader reads all 2D and 3D geometry, as well as texture maps. The hierarchy defined by the keyframe data from keyframe 0 is preserved, including instancing. Entities are positioned based on keyframe 0.

Supported Entities

- All 2D and 3D geometry
- Cameras
- Groups
- Texture maps in the formats: 8-bit color-mapped, 16-bit and 24-bit true color, uncompressed or Run Length Encoded TGA, BMP, JPEG, and LWI (LightWork Image).

NOTE Textures from 3DS files come through as **Presenter** materials, though you should bear in mind that 3DS files contain file names in the 8.3 DOS format only and that various formats are not yet supported in **Presenter**.

- Colors (from material color, not wireframe color - ambient, diffuse, shininess, transparency and self illumination).

Unsupported Entities

- Keyframes (objects are currently taken from keyframe 0)
- Texture maps in the formats: gray-scale TGA, TIF, GIF, and PNG.
- Other maps (for example, opacity maps, reflections, and so on)
- Wireframe meshes
- Lines, splines
- Points
- Background images

See also:

[3DS File Reader Options](#) (page 792)

ASCII Laser Scan File Reader

Most scanner software support exporting the point data in an ASCII text file. Providing that the data is saved in the correct format, this data can be read by Autodesk Navisworks.

Supported formats for ASCII laser scan data are listed below. The data must be separated using one of the following characters: comma, tab or space. The character used to signify a decimal must be a point (period).

- X, Y, Z
- X, Y, Z, Intensity
- X, Y, Z, Red, Green, Blue
- X, Y, Z, Intensity, Red, Green, Blue

Intensity, Red, Green, and Blue values are integers in the range 0-255. Intensities are not gamma corrected.

Supported Entities

- Points

Unsupported Entities

- No other entities are supported.

See also:

[ASCII Laser Scan File Reader Options](#) (page 792)

Bentley AutoPLANT File Reader

Bentley AutoPLANT is based on AutoCAD and as such uses the DWG file format to store model geometry. Any settings related to the DWG file format also affect files from AutoPLANT.

AutoPLANT Object Properties can be stored in external database MDB files. By default, Autodesk Navisworks also supports these files through the DataTools functionality, and looks for Equipment, Nozzle and Piping datatools links.

See also:

[DWG/DXF File Reader Options](#) (page 797)

CIS/2 File Reader

The CIS/2 file reader supports CIMSteel Integration Standards (CIS/2) adopted by the American Institute of Steel Construction (AISC) as their format for data exchange between steel related CAD software.

Supported Entities

- Assembly_design
- Assembly_design_child
- Assembly_manufacturing
- Analysis_model_3d
- Part_derived
- Part_prismatic_simple
- Part_sheet_bounded_complex
- Part_sheet_bounded_simple

- Section_profile_compound
- Section_profile_derived
- Section_profile_edge_defined
- Section_profile_angle
- Section_profile_channel
- Section_profile_circle
- Section_profile_circle_hollow
- Section_profile_i_type
- Section_profile_i_type_asymmetric
- Section_profile_i_type_rail
- Section_profile_rectangle
- Section_profile_rectangle_hollow
- Section_profile_t_type
- feature_volume_prismatic_chamfer
- feature_volume_prismatic_flange_notch
- feature_volume_prismatic_flange_chamfer
- feature_volume_prismatic_notch
- feature_volume_prismatic_skewed_end
- Element_curve_simple
- Element_node_connectivity
- Element_eccentricity
- Located_joint_system
- Design_joint_system
- Joint_system_mechanical
- Fastener_mechanism
- Fastener_simple_bolt
- Fastener_simple_shear_connector

Unsupported Entities

- Part_complex
- Part_prismatic_complex
- Part_sheet_profiled

- Section_profile_centreline
- feature_cutting_plane
- feature_edge_chamfer
- feature_surface
- feature_thread
- feature_volume_complex
- feature_volume_curved
- feature_volume_hole
- Element_volume
- Element_surface
- Element_point
- Element_curve_complex
- Element_with_material
- Joint_system_amorphous
- Joint_system_chemical
- Joint_system_welded
- Weld_mechanism
- Joint_system_complex
- Fastener_simple_nut
- Fastener_simple_washer
- Fastener_simple_stud
- Fastener_simple_pin
- Fastener_simple_nail
- Fastener_simple_screw
- Fastener_simple_countersunk
- Fastener_simple_curve
- Fastener_simple_complex

Supported Basic Section Profile Types

The file reader supports the following basic section profile types:

- I-Beam

- Tee
- Angle
- Channel
- Circle
- Rectangle
- Double Angle
- Joist

Supported AISC Section Profile Types

The reader supports all AISC defined section profile types and maps them to the basic section profile types:

- I-Beam: HP, M, S, W
- Tee: MT, ST, WT
- Angle: L, LP
- Channel: C, MC
- Circle: HSRO, P, PX, PXX, RB
- Rectangle: HSRE, HSSQ, SB, TS
- Double Angle: DL, DLL, DLS

See also:

[CIS2 File Reader Options](#) (page 793)

DWG/DXF File Reader

The DWG/DXF file reader uses Autodesk's ObjectDBX™ technology and so is guaranteed to read all object geometry and information for those third party applications that utilize the ObjectDBX Framework.

The structure of the drawing is preserved including xrefs, blocks, inserts, AutoCAD color index, layers, views and active viewpoint. Entities are colored using the AutoCAD Color Index (ACI), so will match those in an AutoCAD 'shaded' view.

NOTE The file reader supports files from all products based on AutoCAD 2012 and earlier.

Supported Entities

- All 2D and 3D geometry, including arcs, lines, polylines with non-zero thickness, ACIS objects (regions and solids), polygon and polyface meshes, 3D faces and surfaces.
- Points and snap points.
- Lines, polylines, circles, arcs with zero thickness.
- Named views
- Layers
- Colors
- Blocks, inserts and multiple inserts
- Groups
- External references (xrefs)
- Hyperlinks
- Text or multi-line text
- Entity handles
- Attributes
- Textures
- File properties

Unsupported Entities

- Lights
- Splines
- Multi-lines
- Linetypes
- Dimensions and leaders
- Raster bitmaps
- Construction lines (xlines and rays)
- Hatching

See also:

[DWG/DXF File Reader Options](#) (page 797)

Overview of Object Enablers

Object Enablers (OEs) can be thought of as a ‘translator’ for custom objects. Applications based on the AutoCAD® platform, (including Autodesk products like AutoCAD® Architecture, and non-Autodesk products like Bentley AutoPlant) generate custom objects that are saved into the DWG file format. These custom objects are typically found in content libraries that are built into these applications.

Many applications have the ability to read DWG files; however, outside of the authoring application any custom objects cannot be interpreted. As a result, DWG file readers display such custom objects incorrectly, as proxy graphics (wireframe geometry), unless the required OE is installed on the machine reading the DWG file.

NOTE Autodesk Navisworks contains versions of RealDWG™ for 2005, 2006, 2007, 2008, 2009, 2010, and 2012. This means you can use OE between 2005 and 2012 with Autodesk Navisworks. You cannot use any OEs older than 2005. However, OEs are backwards compatible, and you can install a later OE to support reading of older versions of DWG files.

To check if an OE is required

- 1 In Autodesk Navisworks, open a DWG file.
- 2 If you can only see wireframe geometry, click the **Home** tab, slide out the **Project** panel, and click **Scene Statistics**. The **Scene Statistics** dialog box contains the report listing any missing, or failed to load OEs. You need to install all missing OEs to open the file correctly in Autodesk Navisworks.

NOTE Occasionally, an OE is already installed, but does not support non-AutoCAD-based applications, like Autodesk Navisworks. In such cases, the OE is reported as missing in the **Scene Statistics** dialog box.

To install and use an OE with Autodesk Navisworks

- 1 Obtain the required OE, and run the installer.
- 2 On the OE Setup page, select the check box for the relevant Autodesk Autodesk Navisworks products.

NOTE Some OEs have dependencies on other OEs, and, therefore, must be installed in a specific order. For example, the AutoCAD MEP 2009 OE has a dependency on the AutoCAD Architecture 2009 OE. As a result, the AutoCAD MEP 2009 OE must be installed after the AutoCAD Architecture OE.

- 3 Click **Install**. When the OE is installed, click **Finish**.
- 4 Run Autodesk Navisworks.
- 5 Click the application button  **> Options**.
- 6 In the **Options Editor**, expand the **File Readers** node, and click **DWG/DXF**.
- 7 Ensure the value in the **DWG Loader Version** field is the same as the version of the installed OE, and close the dialog box. For example, if you've installed OE for AutoCAD Architecture 2007, you must set the DWG Loader Version to '2007'.

When you open a DWG file in Autodesk Navisworks, the file reader uses the configured OE.

How do I know if I need to use an Object Enabler with Autodesk Navisworks2012?

When a DWG file is opened in Autodesk Navisworks2012, and the objects are displayed as wireframe geometry, it usually means that an Object Enabler is missing.

Additionally, the **Scene Statistics** dialog box reports any missing or failed to load OEs.

Where do I obtain Object Enablers from?

The recommended way to obtain OEs is to request them from the persons authoring the DWG file. This ensures that the correct version of OE is used.

TIP You can also download many OEs from the vendor websites for the authoring applications. For example, you can download Autodesk OEs from <http://www.autodesk.com/oe>.

What do I do when Object Enablers are not reported as missing but I can only see wireframe geometry?

- 1 Click the application button  **> Options**.

- 2 In the **Options Editor**, expand the **File Readers** node, and click **DWG/DXF**.
- 3 Select **Shaded** from the **Render Type** drop-down list.
- 4 Ensure the **Use ADT Standard Configuration** check box is selected.
- 5 Click **OK** to save the changes and close the dialog box.
- 6 Click **New**  on the **Quick Access** toolbar, and then open the DWG file again.

DWF/DWFX File Reader

Autodesk DWF was specifically developed by Autodesk as a file format for architects, engineers, and GIS professionals to share 2D- and 3D- design data. DWF files are highly compressed and retain detailed design information and scale. The newest version of the DWF file format, DWFX, is based on the XML Paper Specification (XPS) from Microsoft. DWFX files can be opened and printed instantly using the free Microsoft XPS Viewer, which comes pre-installed on computers using the Microsoft Windows Vista® operating system. (For the Windows XP operating system, the Microsoft XPS Viewer can be downloaded directly from Microsoft.) Unlike DWF files, DWFX files include additional information to display design data in the Microsoft XPS Viewer. As such, DWFX files are larger than corresponding DWF files.

Supported Entities

- All 3D geometry
- Texture maps
- Texture coordinates
- Colors (per-vertex, per-face)
- Property fields
- Categories
- 2D lines/plot sections
- Thumbnails (for 2012 files or later)
- Sheet property (for 2012 files or later)
- More than one 3D section per file (multi-sheet file support)

Unsupported Entities

- Marked-up sketches
- NURBS surfaces
- Cameras

See also:

[DWF File Reader Options](#) (page 796)

DGN File Reader

Autodesk Navisworks can read 3D DGN and PRP files from Bentley's MicroStation, but does not support CEL files or 2D DGN files. Referenced files and instances of cells are respected, and the **Selection Tree** reflects this file structure.

NOTE The file reader supports files from MicroStation 95, SE and /J. It does not support MicroStation Modeler and any versions of MicroStation before 95.

Supported Entities

- All 2D and 3D geometry including shapes, complex shapes, meshes, cones, surfaces, B-spline boundaries, solids, SmartSolids and Feature Solids, lines, arcs and ellipses.
- Splines and B-spline curves
- Lights
- Levels
- Cells, shared cells, and their instancing
- Colors and ambient, diffuse, and shininess properties of materials from PAL and MAT palette and material files
- Texture maps
- Referenced files including aliases
- Dynamic drawing of parametric models when loading/exporting DGN and PRP files
- 3D text used for notes and labels is now converted and displayed by default

- Family, part and texture information from TriForma, and PDS object information from DRV files
- Family, part and texture information from TriForma, and PDS object information from DRV files

Unsupported Entities

- Raster bitmaps
- Dimensions and leaders

See also:

[DGN File Reader Options](#) (page 793)

Faro Scan File Reader

The file reader supports files from all Faro™ scanners. Combined iQscan files must be located in a folder called 'Scans' in the same directory as the associated iQmod and iQwsp workspace files.

See also:

[Faro Scan File Reader Options](#) (page 801)

FBX File Reader

The file reader supports Autodesk FBX files.

The Autodesk FBX format is a free platform-independent 3D authoring and interchange format that provides access to 3D content from most 3D vendors. FBX file format offers you improved interoperability between Autodesk Navisworks and a range of Autodesk products including Maya, 3DS MAX, and SoftImage.

The Autodesk Navisworks FBX file reader reads all 2D and 3D geometry, as well as texture maps and material maps. However, currently it does not support animation.

Supported Entities

- All 2D and 3D geometry (mesh, NURBS, patch, trimmed NURBS, NURBS curve)
- Texture maps
- Material maps
- Camera and light
- Skeleton
- Normals, colors (they are both from vertex)

Unsupported Entities

- Constraint (cluster constraint, shape constraint and aim constraint)
- Pose
- Take of animation
- Marker
- Nulls
- Point Cloud geometry

See also:

[FBX File Reader Options](#) (page 801)

IFC File Reader

The file reader supports stand-alone IF files.

Supported Entities

- Faceted BReps
- Extruded area solids
- Geometric sets
- Face-based and surface-based models
- Simple, trimmed, and composite curves
- Simple surfaces
- Simple parametric, arbitrary and derived profiles

- Boolean clipping results and element-level voiding and projection CSG operations
- Basic styled and mapped items
- Property sets, including simple and complex properties

Unsupported Entities

- Voided BReps
- Bounded half-space solids
- Complex parametric profiles
- BSpline curves
- Curve styles
- Swept surfaces
- Textures and complex lighting
- Loading of server-based IFC models

See also:

[IFC File Reader Options](#) (page 802)

IGES File Reader

The file reader uses Inventor Server to load IGS and IGES files. All file versions are supported.

You cannot customize the IGES file reader in the **Options Editor** any longer.

Inventor File Reader

The file reader supports IPT (part), IAM (assembly) and IPJ (project) file formats. IDW (drawing) file format cannot be read.

The reader supports files from Autodesk® Inventor® 2012 and earlier.

See also:

[Inventor File Reader Options](#) (page 803)

JTOpen File Reader

The file reader supports 3D JT data format developed by Siemens PLM Software (formerly UGS Corp.).

Supported Entities

- Tri-strip set shape
- Polyline set shape
- Box primitive shape
- Cylinder primitive shape
- Sphere primitive shape
- Models with vertex-based color and normal
- General JT B-Rep
- Geometric transform attribute
- Material attribute
- XT B-Rep segment
- Properties

Unsupported Entities

- Point set shape
- Polygon set shape
- Wire harness set shape
- Pyramid primitive shape
- Tri-prism primitive shape
- PMI manager meta data
- Models with facet-based or primitive-based color and normal
- Texture
- HSV color model
- B-Rep CAD tag
- Line style attribute
- Point style attribute
- Shader effects attribute

- Vertex shader attribute
- Fragment shader attribute
- Infinite light attribute
- Point light attribute
- Wireframe segment
- LOD

See also:

[JTOpen File Reader Options](#) (page 805)

Leica Scan File Reader

The file reader supports files from all Leica™ HDS scanners.

Supported Entities

- Points

Unsupported Entities

- No other entities are supported

See also:

[Leica Scan File Reader Options](#) (page 804)

MAN File Reader

The file reader supports MAN files from Informatix's MicroGDS™ version 6.0 or later. MicroGDS™ projects are not supported. The workaround is to export the desired project window with a .man file extension.

In Autodesk Navisworks, MicroGDS renderer materials are shown in their flat-shaded colors in shaded mode. In full render mode, or with **Presenter** rendering, the full shaders are used. Only the standard LightWorks shaders are available. The shaders which are unique to MicroGDS are not available inside Autodesk Navisworks, and are converted as follows:

- Height band color shader is treated as plain grey

- Wrapped random color shader is treated as a plain color using the flat-shaded color from MicroGDS
- Wrapped stencil transparency is ignored
- Undulate, wrapped brick, wrapped grid and wrapped ripple displacement shaders are ignored
- Object axis texture space is equivalent to the Autodesk Navisworks box texture space
- Auto axis and object XY axis texture spaces are treated as the Autodesk Navisworks box texture space
- Grid background is treated as a plain background using the background color, that is grid lines are not shown
- Foreground and environment shaders are ignored

NOTE All other shaders, as of MicroGDS 7.2, are correctly imported into Autodesk Navisworks.

MicroGDS materials are specified in millimeters, and are converted into metres to make Autodesk Navisworks materials, dividing distance parameters by 1000.

Windows with Perspective Views are converted as viewpoints.

Supported Entities

- Clump primitives
- Line primitives.

NOTE The color of line primitives is determined by the first phase in which they appear in the Principal window of the MAN file. If they are not included in the Principal window, the color is determined by their style.

- Light styles.

NOTE Projector lights are treated as a spot light without the transparent image.

- Material styles, both plain and most LightWorks Renderer materials.

NOTE For materials using wrapped images, you must specify the texture paths.

- Layers.

NOTE All layers are read, and made visible according to their status in the Principal window of the MAN file.

- Instances
- Object data structure

Unsupported Entities

- Text primitives
- Photo primitives

See also:

[MAN File Reader Options](#) (page 805)

Parasolid File Reader

The file reader supports X_B Parasolid files.

See also:

[Parasolid File Reader Options](#) (page 806)

PDS File Reader

The file reader supports DRI files from the PDS Design Review package.

See also:

[PDS File Reader Options](#) (page 806)

Pro/ENGINEER File Reader

The Pro/ENGINEER file reader uses Inventor Server to load the following types of Pro/ENGINEER files:

- *.prt* (part) - up to version Wildfire 5.0
- *.asm* (assembly) - up to version Wildfire 5.0
- *.g (Granite) - up to version 6.0
- *.neu* (Neutral)

The file reader uses Autodesk Inventor Server to translate assembly and part files, solids, multi-solids, surfaces, and more. You cannot customize the Pro/ENGINEER file reader in the **Options Editor**.

Riegl Scan File Reader

The file reader supports files from all Riegl™ LMS scanners.

Supported Entities

- Points
- Triangles

Unsupported Entities

- No other entities are supported

See also:

[Riegl Scan File Reader Options](#) (page 807)

RVM File Reader

The file reader supports the following file formats:

- Binary and ASCII RVM files exported from AVEVA's PDMS™ product
- Dump Attributes and Data (also called "OUTPUT") attribute files, with the .att, .attrib, and .txt file extensions.
- RVS files

Supported Entities

- All geometry
- Attributes stored on groups
- Textures (via RVS file)
- Cameras and camera tracks (via RVS file)
- Clip planes (via RVS file)
- Signs (via RVS file)

- Tags (via RVS file)
- Labels (via RVS file)
- Translucency (via RVS file)
- PDMS origin points

Unsupported Entities

- Attributes stored on primitives
- Lights
- Object Animation
- Smooth Animation
- Groups
- Autotags

See also:

[RVM File Reader Options](#) (page 808)

SAT File Reader

The file reader supports ACIS SAT files.

See also:

[SAT File Reader Options](#) (page 809)

SketchUp SKP File Reader

The file reader supports native SKP file format.

Supported Entities

- Geometry
- Materials (face front material only)
- Transparency
- Groups

- Components
- Layers
- Imported images
- Transparency

Unsupported Entities

- Text
- Dimensions
- Section planes

See also:

[Sketchup SKP File Reader Options](#) (page 810)

STEP File Reader

The file reader uses Inventor Server to load STP and STEP files up to and including AP214 CC2 and AP203.

You cannot customize the STEP file reader in the **Options Editor** any longer.

STL File Reader

The file reader only supports binary STL files. ASCII versions are not supported.

Supported Entities

- Triangles

Unsupported Entities

- No other entities are supported

See also:

[STL File Reader Options](#) (page 811)

VRML File Reader

The file reader supports files in both VRML1 and VRML2 file formats.

Supported Entities

- All 3D geometry including cuboids, cylinders, cones, spheres, elevation grids, extrusions, face and line sets, and points.
- All grouping nodes, but with the following limitations:
 - VRML2 Billboard nodes - children are loaded but no billboarding takes place.
 - VRML2 Collision nodes - children are loaded but no specification of collision detection occurs.
 - VRML1 WWWAnchor and VRML2 Anchor nodes - children are loaded but clicking objects does not load any referenced VRML world.
 - VRML1 and VRML2 LOD nodes - the most detailed (that is, first) child is always loaded.

Unsupported Entities

- All ROUTE definitions
- All sensor nodes
- All interpolator nodes
- Textures specified within the VRML file (VRML2 PixelTexture nodes and the image component of VRML1 Texture2 nodes).
- VRML2 Script nodes
- VRML2 MovieTexture nodes
- VRML2 Fog nodes
- VRML2 AudioClip and Sound nodes
- All text-related nodes (VRML1 ASCII Text, VRML2 Text, and FontStyle nodes).

See also:

[VRML File Reader Options](#) (page 811)

Z+F Scan File Reader

The file reader supports files from all Z+F™ IMAGER scanners.

Supported Entities

- Points

Unsupported Entities

- No other entities are supported

See also:

[Z+F File Reader Options](#) (page 812)

Use File Exporters

Autodesk Navisworks provides file exporters to create native Autodesk Navisworks files directly in CAD applications.

Currently, you can export NWC files from AutoCAD, MicroStation, Revit, ArchiCAD, and Viz/Max applications.

You can use file exporters:

- If Autodesk Navisworks cannot read the native CAD file format. Currently, the following native file formats are not supported: VIZ, MAX, ArchiCAD, Revit.
- If Autodesk Navisworks has converted the native CAD file, but some geometry is missing, for example, and you want to improve the file quality.

AutoCAD File Exporter

Autodesk Navisworks comes with ARX plugins for any AutoCAD™ based product, such as Architectural Desktop™, and enables you to export files into NWC file format.

The file exporter is available for any AutoCAD-based product between AutoCAD 2004 and 2012 releases.

NOTE If you installed the Autodesk Navisworks software with exporter plugins, and you install any AutoCAD-based software after Autodesk Navisworks is installed, you need to update your exporter plugins installation with the appropriate ARX plugins before you can use them.

See also:

[DWG File Exporter Options](#) (page 813)

Add the ARX Plugin

You can either add the ARX plugin to Exporter Plugins installation, or load it manually in AutoCAD-based software.

To add the ARX plugin to Exporter Plugins installation

- 1 In the **Control Panel**, double-click **Add or Remove Programs**.
- 2 In the **Add or Remove Programs** dialog box, click Autodesk Navisworks Manage 2012 Exporter Plugins, and then click **Change/Remove** in Windows XP or **Uninstall/Change** in Vista/Windows 7.
- 3 In the Installation wizard, click **Add or Remove Features**.
- 4 On the **Select Exporters to Install** page, the **Export From Other Applications** node, select the check box next to the required plugin.
- 5 Click **Next**.
- 6 On the Autodesk Navisworks Manage 2012 **Update Exporter Plugins** page, click **Next**.
- 7 On the **Update Complete** page, you are informed when the updates have been performed. Click **Finish**.

To load the ARX plugin in an AutoCAD-based application

- 1 Type **ARX** at the command line.
- 2 Press Return.
- 3 Type the letter **L** (for 'Load') at the command line.
- 4 Press Return.
- 5 In the **Select ARX/DBX File** dialog box, browse to the ARX plugin.

NOTE For AutoCAD 20xx based applications, the plugin is located under *C:\Program Files\Common Files\Autodesk Shared\Autodesk Navisworks\2012\NWExport20xx\nwexport20xx.arx*. So, for example, for AutoCAD 2004 based applications, the plugin is located under *C:\Program Files\Common Files\Autodesk Shared\Autodesk Navisworks\2012\NWExport2004\nwexport2004.arx*, and so on.

- 6 Click **OK** to load the ARX plugin.

Use the ARX Plugin

You can export files from AutoCAD-based applications either from the command line, or by using the Autodesk Navisworks menu.

Before you can use the Autodesk Navisworks menu, you need to load it first. By default, the Autodesk Navisworks menu is added to the menu browser. In some programs, it can also be added to the menu bar.

To use the command line to export files

- 1 To export an NWC file, type **NWCOUT** at the command line.
- 2 Press Return.
- 3 In the **Save As** dialog box, enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 4 Click **OK** to export the file, or **Cancel** to return to the application without exporting it.

To load the Autodesk Navisworks export menu

- 1 At the command line, type **MENULOAD**.
- 2 Press Return.
- 3 In the **Load/Unload Customizations** dialog box, click **Browse**.
- 4 In the **Select Customization File** dialog box, change the **Files of Type** to **Legacy Menu Template (*.mnu)**, and browse to the Autodesk Navisworks menu file.

NOTE For AutoCAD 20xx based applications, this file is located under *C:\Program Files\Common Files\Autodesk Shared\Autodesk Navisworks\2012\NWExport20xx\lwnw_export.mnu*. So, for example, for AutoCAD 2004 based applications, this file is located under *C:\Program Files\Common Files\Autodesk Shared\Autodesk Navisworks\2012\NWExport2004\lwnw_export.mnu*, and so on.

- 5 Click **Open**.
- 6 In the **Load/Unload Customizations** dialog box, click **Load**.
The loaded menu is shown in the **Loaded Customization Groups**.
- 7 Click **Close**.

To use Autodesk Navisworks menu to export files

- 1 Click the menu browser button located at the top-left corner of the application window.
- 2 To export an NWC file, click **Autodesk Navisworks Cache** on the Autodesk Navisworks menu.
- 3 In the **Save As** dialog box, enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 4 Click **OK** to export the file, or **Cancel** to return to the application without exporting it.

To adjust the options for the DWG file exporter

- 1 Click the menu browser button located at the top-left corner of the application window.
- 2 Click **Autodesk Navisworks Export Option** on the Autodesk Navisworks menu.
- 3 Expand the **File Exporters** node in the **Options Editor**, and click the **DWG** page. Use the options on this page to adjust the settings for future exports of NWC files from AutoCAD-based applications.
- 4 Click **OK** to save the changes.

 **Command entry:** NWOPT

CAD Preview

The ARX plugin also enables you to walk through your model in real time inside a dockable Navigator window directly in the AutoCAD interface.

You can easily import and export viewpoints between Navigator and AutoCAD, and explore the models that you are building.

NOTE The Autodesk Navisworks Navigator is provided for products based on AutoCAD 2004™ and above.

To navigate AutoCAD models in real time

- 1 Type **NWNAVIGATOR** at the command line.
- 2 Use the Navigator window to explore your model.

Buttons

The Navigator toolbar contains the following buttons.

Button	Description
	Synchronizes the model in the Navigator window with the AutoCAD model. The Navigator window is not updated automatically. Click this button every time you want to navigate around the latest model.
	Changes the current AutoCAD view to the view in the Navigator window.
	Changes the view in the Navigator window to the current AutoCAD view.
	Selects walk mode and enables you to walk through the model on a horizontal plane.
	Selects look around mode and enables you to look around the model from the current camera position.
	Selects zoom mode. Dragging the left mouse button up and down or using the UP and DOWN ARROW keys zooms in and out respectively.

Button	Description
	Selects zoom-to-a-box mode. Dragging a box with the left mouse button over the model fills the view with the contents of the box. Holding down the SHIFT or CTRL keys or spinning the mouse wheel, temporarily switches this mode to zoom mode.
	Selects pan mode. Dragging the left mouse pans the model up, down, left and right. Holding down the SHIFT or CTRL keys or spinning the mouse wheel, temporarily switches this mode to zoom mode.
	Selects orbit mode. Dragging the left mouse button or using the UP, DOWN, LEFT ARROW, and RIGHT ARROW keys rotates the camera around the model. Holding down the SHIFT key or spinning the mouse wheel, temporarily switches this mode to zoom mode. Holding down the CTRL key temporarily pans the camera.
	Selects examine mode. Dragging the left mouse button or using the UP, DOWN, LEFT ARROW, and RIGHT ARROW keys rotates the model about. Holding down the SHIFT key or spinning the mouse wheel, temporarily switches this mode to zoom mode. Holding down the CTRL key temporarily switches this mode to pan mode.
	Selects fly mode. Holding down the left mouse button moves the camera forward. As in a flight simulator, the left mouse button banks left/right when dragged left or right and tilts up/down when dragged up or down. The UP and DOWN cursor keys zoom in and out respectively, and the LEFT ARROW and RIGHT ARROW keys spin the

Button	Description
	camera left and right respectively. Holding down the SHIFT key speeds up this movement. Holding down the CTRL key rotates the camera around its viewing axis, while still moving forward.
	Selects turntable mode. Dragging the left mouse button left and right, or using the LEFT ARROW and RIGHT ARROW keys spins the turntable left and right respectively. Holding down the SHIFT key or spinning the mouse wheel, temporarily switches this mode to zoom mode. Holding down the CTRL key temporarily switches this mode to pan mode.
	Shows all model. Clicking this button dollies and pans the camera so that the entire model is displayed.
	Selects perspective camera view.
	Selects orthographic camera view.
	Opens the Help system.

The Shortcut Menu

Right-clicking in the Navigator window opens a shortcut menu.

Focus on Item Focuses the camera on the selected item. The point you click becomes the focal point for examine, orbit, and turntable navigation modes.

Speed

Frame Rate Specifies the number of frames per second (FPS) that are rendered in the Navigator window.

The default setting is 6. You can set the frame rate from 1 through 20 frames per second. Reducing the value reduces drop-out, but can cause jerky movement during navigation. Increasing the value ensures a smoother navigation, but increases drop-out.

Hardware Acceleration Select this option to utilize any available OpenGL hardware acceleration on your video card.

NOTE If your video card does not support OpenGL hardware acceleration, this option is not available.

Viewpoint

Rendering Selects rendering mode for your model. Choose from:

- **Shaded** - renders the model with smooth shading and without textures. This is the default option.
- **Wireframe** - renders the model in wireframe; all triangle edges are visible in this mode.
- **Hidden Line** - renders the model in hidden line; this is equivalent to rendering model as shaded and wireframe at the same time. The output is low quality as all the facet edges in the model are visible.

Lighting Selects lighting mode for your model. Choose from:

- **No Lights** - turns off lighting. The model is shaded with flat rendering.
- **Head Light** - uses a single directional light located at the camera that always points in the same direction as the camera. This is the default option.
- **Scene Lights** - uses any lights defined in the model, or two default opposing lights, if none are available.

Navigation Mode Selects navigation mode. The options available here are the same as on the **Navigator** toolbar, with one exception. The **Select** option enables you to select items in your model. You can select multiple items by holding down the CTRL key while selecting items.

Navigation Tools Select **View All** to display the entire model.

Perspective Camera Selects perspective camera view.

Orthographic Camera Selects orthographic camera view.

Options

Culling Options Opens the Culling Options Dialog box.

Close Files on Open Indicates whether NWC file created for the model is closed once it has been loaded into memory.

Selecting this option unlocks NWC files for editing by other users.

Max Image Texture Size Specifies the maximum size for texture images in pixels.

The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Revit File Exporter

Autodesk Navisworks cannot read native Revit files directly. Use the file exporter to save your files in NWC format, which can be open in Autodesk Navisworks.

The file exporter is available for Revit versions 9.0 to 2012.

See also:

[Revit File Exporter Options](#) (page 817)

To export NWC files from Revit

- 1 In Revit, click **Tools** ► **External Tools** ► Autodesk Navisworks2012.

NOTE This option is not available in Demo/Viewer mode. If you are not in demo mode, but do not have access to the Autodesk Navisworks menu, check if editing view is set to normal, and the modify tool is selected (► **Modify**).

- 2 In the **Export Scene As** dialog box, enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 3 Click **Save** to export the file, or **Cancel** to return to the application without exporting it.

To adjust the options for the Revit file exporter

- 1 In Revit, click Tools ► External Tools ► Autodesk Navisworks2012.
- 2 In the **Export Scene As** dialog box, click the Autodesk Navisworks **Settings** button.

- 3 Expand the **File Exporters** node in the **Options Editor**, and click the **Revit** page. Use the options on this page to adjust the settings for future exports of NWC files from Revit.
- 4 Click **OK** to save the changes and return to the **Export Scene As** dialog box.
- 5 Click **Cancel** to close the dialog box.

MicroStation File Exporter

Autodesk Navisworks comes with MDL plugins for MicroStation™ J, v8, and v8.9, and enables you to export files into NWC file format.

NOTE If you installed the Autodesk Navisworks software with exporter plugins, and you install MicroStation software after Autodesk Navisworks is installed, you need to update your exporter plugins installation with the appropriate ARX plugins before you can use them.

See also:

[DGN File Exporter Options](#) (page 819)

To add the MDL plugin to Autodesk Navisworks Exporter Plugins installation

- 1 In the **Control Panel**, double-click **Add or Remove Programs**.
- 2 In the **Add or Remove Programs** dialog box, click Autodesk Navisworks Manage 2012 Exporter Plugins, and then click **Change/Remove** in Windows XP or **Uninstall/Change** in Vista/Windows 7.
- 3 In the Installation wizard, click **Add or Remove Features**.
- 4 On the **Select Exporters too Install** page, the **Export From Other Applications** node, select the check box next to the required plugin.
- 5 Click **Next**.
- 6 On the Autodesk Navisworks Manage 2012 **Update Exporter Plugins** page, click **Next**.
- 7 On the **Update Complete** page, you are informed when the updates have been performed. Click **Finish**.

Load the MDL Plugin

Before you can use the MDL plugin, you need to load it first.

You can either do it for the current session only, or you can add the plugin to your default configuration, in which case, it is loaded for all future sessions.

To load the MDL plugin for the current session only

- 1 Click **Utilities > Key-In**.
- 2 In the **Key-In** dialog box, type **mdl load nwexport9**.
- 3 Press Return.

To load the MDL plugin for all future sessions

- 1 Click **Workspace > Configuration**.
- 2 In the **Configuration** dialog box, **Category** field, click **Design Applications**.
- 3 Click **NWEXPORT9** in the **Available Applications** field.
- 4 Click the **Add** button.
- 5 Click **OK**, and confirm that you want to add NWEXPORT MDL plugin to your default configuration.

Export Files from the Key-In Command Line

You can export files from the key-in command line individually or in batches.

You can also add the commands to the **Key-In** menu to speed up the process.

To use the key-in command line to export files

- 1 Click **Utilities > Key-In** to open the **Key-In** dialog box.
- 2 To export an NWC file, type **nwcout**.

NOTE To export the current design file quickly, you can use the **batchnwcout** command. The explorer replaces the design file extension (usually .dgn) with .nwc when exporting the file. When you use this command, you are not prompted to overwrite any existing files, and no export dialog box is shown.

- 3 In the Autodesk Navisworks **Export File** dialog box, browse to the desired storage location.

The exporter automatically provides a filename, and a view number, which you can modify, if necessary. The chosen view number determines the initial model view in Autodesk Navisworks.

- 4 Click **OK** to export the file.

To add Autodesk Navisworks export commands to the Key-In menu

- 1 Click **Workspace ► Customize**.
- 2 In the **Customize** dialog box, the **Menu Bar** tab, expand the **Utilities** option in the **Edit Menu Bar** field.
- 3 Click the **Key-In** option, and click the **Insert** button.
- 4 In the **Insert Menu Item** dialog box, enter the **Label**, for example, "Export Autodesk Navisworks Cache File".
- 5 In the **Key-In** field, type the appropriate command, for example, "nwcout".
- 6 Click **OK**.
- 7 Add as many menu items as necessary, and click **Save**.

The added options are now available from the **Utilities ► Key-In** menu.

Export Files from the Command Line

You can also export files from the command line by using the msbatch.bat file. This is useful when you want to set up an automated script for file export.

NOTE To export a file from the command line, you must first add the MicroStation program directory to the Path environment variable.

The format of the command line exporter is

```
msbatch nwexport9 [-f] FILE [OUTPUT]
```

FILE is the name of the MicroStation file you want to export. This is the only required option. The **OUTPUT** argument enables you to specify the name of the output file.

By default, the exporter only converts files that have been changed since the last time you exported them. This is useful for speeding up exporting a large

number of files. If you want to force the exporter to convert all files, use the **-f** option.

NOTE If the exporter encounters any problems, it produces an error log in a file called **nwdout.err**.

To modify the Path environment variable

- 1 In the **Control Panel**, double-click **System**.
- 2 In the **System Properties** dialog box, **Advanced** tab, click the **Environment Variables** button.
- 3 In the **Environment Variables** dialog box, double-click **Path** under **System Variables**.
- 4 In the **Edit System Variable** dialog box, add the path to the MicroStation application to the **Variable Value** field.

NOTE Paths must be separated with a semicolon (;).

- 5 Click **OK** to save the changes and close the dialog boxes.

To use the command line to export files

- 1 On the **Start** menu, click **Run** in Windows XP or press Windows + R on the keyboard in Vista and Windows 7.
- 2 Enter the file export command with appropriate arguments, for example **msbatch nwexport9 file**.
- 3 Press Enter.
- 4 In the **MicroStation Manager** dialog box, select the files you want to export, file extension (NWC), and storage location.
- 5 Click **OK**.

Customize the DGN File Exporter Options

The file exporter only exports from 3D DGN files - 2D files are not supported.

Autodesk Navisworks colors are derived from either MicroStation cell colors or MicroStation materials, depending on the chosen export options. The appearance of objects in Publisher matches the appearance of a MicroStation shaded render.

To adjust the options for the DGN file exporter

- 1 Click **Utilities ► Key-In** to open the **Key-In** dialog box.
- 2 Type **nwopt**, and press Enter.
- 3 Expand the **File Exporters** node in the **Options Editor**, and click the **DGN** page. Use the options on this page to adjust the settings for future exports of NWC files from MicroStation.
- 4 Click **OK** to save the changes.

Viz and Max File Exporter

Autodesk Navisworks cannot read native Viz/Max files directly. Use the file exporter to save your files in NWC format, which can be open in Autodesk Navisworks.

The file exporter is available for Viz versions 2007 and 2008 and Max versions from 8 to 2012.

NOTE If you installed the Autodesk Navisworks software with exporter plugins, and you install Viz or Max software after Autodesk Navisworks is installed, you need to update your exporter plugins installation with the appropriate ARX plugins before you can use them.

See also:

[Viz/Max File Exporter Options](#) (page 821)

To export NWC files from Viz or Max

- 1 Click **File ► Export**.
- 2 In the **Select File to Export** dialog box, select Autodesk Navisworks2012 **Cache (*.nwc)** in the **Save as Type** field.
- 3 Enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 4 Click **Save** to export the file, or **Cancel** to return to the application without exporting it.

NOTE Any textures applied to the Viz or Max model are saved in a directory with the same name as the exported file, but with a **_presenter_maps** suffix. All textures are converted into BMP files and saved into this directory for use with Autodesk Navisworks.

To add the Autodesk Navisworks utility menu

- 1 Click the **More** button on the **Utilities** panel.
- 2 In the **Utilities** dialog box, select Autodesk Navisworks2012 option, and click **OK**.

To customize the 3DS file exporter options

- 1 Click the **Options Editor** button on the Autodesk Navisworks menu on the **Utilities** panel.
- 2 Expand the **File Exporters** node in the **Options Editor**, and click the **Viz/Max** page. Use the options on this page to adjust the settings for future exports of NWC files from Viz and Max applications.

ArchiCAD File Exporter

Autodesk Navisworks cannot read native ArchiCAD files directly. Use the file exporter to save your files in NWC format, which can be open in Autodesk Navisworks.

The file exporter is available for ArchiCAD v9 through to v13.

The export add-on for ArchiCAD is available from both the 2D and 3D windows. All standard ArchiCAD elements and library parts can be exported provided they have a 3D representation, and any others are ignored. The file exporter saves both standard materials and custom GDL script materials.

NOTE Only visible layers are exported.

ArchiCAD cutaway plane settings does not set the Autodesk Navisworks section plane; it is used to export items that are physically reduced by the plane.

Supported Entities

- Global Unique Identifiers (GUIDs)
- Custom parameters for library parts defined by GDL scripts
- Storeys
- Library part instances
- Cameras
- Hotlinks

- Sun attributes
- Materials

Unsupported Entities

- Section planes
- Points
- Lines
- Textures

To export from 2D window

- 1 Click **File > Save As**.
- 2 In the **Save As** dialog box, select Autodesk Navisworks2012 (*.nwc) in the **Save As Type** field.
- 3 Enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 4 Click **Save** to begin the export process.
- 5 In the **Export** dialog box adjust the file exporter options:
 - **Export GUIDs** - select this check box to attach a Globally Unique Identifier as a property to each item in the model. In Autodesk Navisworks, GUIDs are used by the **Clash Detective** to track clashes.
 - **Enable Library Part Instancing** - select this check box to make instances of library parts rather than creating new items. This is only possible when multiple library part elements within the ArchiCAD model have exactly the same properties. Instancing these parts means a smaller NWC file, and shorter export times.

NOTE In sectioned models, when this check box is selected all instances show the same sectioning as the original item (usually, the first library part in the file). If you get some unexpected effects, clear this check box.

- **Export Library Part Parameters** - select this check box to save library part parameters (including the user-defined custom parameters) as item properties in Autodesk Navisworks.
- **Export Current Story Only** - select this check box to export the current story only.

Clear this check box to export the whole model. The default view is determined from the bounding box of the model.

- 6 Click **OK**.

To export from 3D window

- 1 Click **File ► Save As**.
- 2 In the **Save As** dialog box, select Autodesk Navisworks2012 (*.nwc) in the **Save As Type** field.
- 3 Enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
- 4 Click **Save** to begin the export process.
- 5 In the **Export** dialog box adjust the file exporter options:

- **Export GUIDs** - select this check box to attach a Globally Unique Identifier as a property to each item in the model. In Autodesk Navisworks, GUIDs are used by the **Clash Detective** to track clashes.
- **Enable Library Part Instancing** - select this check box to make instances of library parts rather than creating new items. This is only possible when multiple library part elements within the ArchiCAD model have exactly the same properties. Instancing these parts means a smaller NWC file, and shorter export times.

NOTE In sectioned models, when this check box is selected all instances show the same sectioning as the original item (usually, the first library part in the file). If you get some unexpected effects, clear this check box.

- **Export Library Part Parameters** - select this check box to save library part parameters (including the user-defined custom parameters) as item properties in Autodesk Navisworks.
- **Export Current Story Only** - this option is not used; everything contained within the view is exported.

When saving from the 3D window, the view becomes the default Autodesk Navisworks view (including window settings, such as cutaway planes).

- 6 Click **OK**.

Manage Files

Open Files

To open files in Autodesk Navisworks, you can either use a standard **Open** dialog box or drag and drop files directly into the **Selection Tree** window.

NOTE If the chosen file is a CAD or laser scan file, Autodesk Navisworks automatically uses an appropriate file reader to open it, provided this file format is supported.

Autodesk Navisworks keeps a list of recently opened files (by default, up to 4 files are shown). You can open any of these files by clicking the application



button. If you want to modify the size of this list, use the **Options Editor (General node > Environment page)**.

You can use the SHIFT and CTRL keys to open several files at the same time. This automatically creates a new “Untitled” Autodesk Navisworks file with the selected files appended together.

For NWD files, it is possible to publish them to a web server, and then open them directly from within Autodesk Navisworks. You can start navigating the model even before the file has been fully downloaded. For this, 10 - 50% is usually sufficient. The greater the hierarchical structure of the model, the closer to 50% download is required. Similarly, the lesser the hierarchical structure of the model, the sooner you can begin the navigation.

NOTE Encrypted DWF files, such as files with Password, and Print/Measure protected files are not currently supported.

To open a file

- 1 Click the application button  > **Open** > **Open** .
- 2 In the **Open** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to the folder where your file is located.
- 3 Select the file, and click **Open**.

 **Toolbar:** Classic user interface: **Standard** > **Open** 

 **Command entry:** CTRL + O

To open NWD files located on a web server

- 1 Click the application button  ► **Open** ► **Open URL** .
- 2 Enter the file address, and click **OK**.

Create Files

When you start Autodesk Navisworks, a new “Untitled” Autodesk Navisworks file is automatically created for you. The new file uses default settings defined in the **Options Editor**, and in the **File Options** dialog box. You can customize these settings, as necessary.

If you have a Autodesk Navisworks file already open, and want to close it and create another file, click **New**  on the **Quick Access** toolbar.

 **Toolbar:** Classic user interface: **Standard** ► **New** 

Save and Rename Files

When you save a Autodesk Navisworks file, you have a choice between an NWD and NWF file formats.

As a rule of thumb, use an NWF file format to save the scene you created by bringing all model files together, and an NWD file format when you simply want a snapshot of your current work.

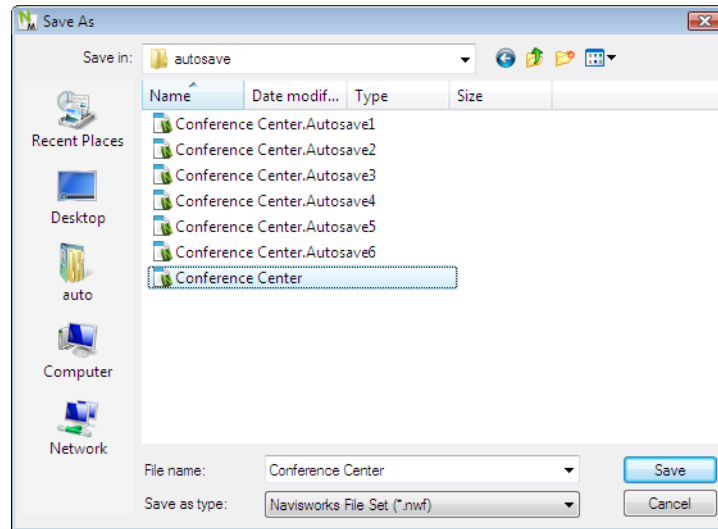
Both formats store the review markups, but NWD file stores the file geometry, while NWF file stores the links to original files. This makes NWF files considerably smaller in size. Also when you open an NWF file, Autodesk Navisworks automatically reloads all modified referenced files, which means the geometry is always up-to-date, even for the most complex models.

On the other hand, whenever you need to share the scene you created and review markups with others, it is best to distribute a published NWD file, which includes additional features such as password protection and file expiration date. The published files can be viewed in both Autodesk Navisworks Manage 2012 and Autodesk Navisworks Freedom 2012 (a free viewer).

IMPORTANT When you publish an NWD file, you cannot include any RPCs added to your scene.

To save a file

- 1 Click **Save**  on the **Quick Access** toolbar. If your file has been saved previously, Autodesk Navisworks overwrites the file with the new data.
- 2 If your file has not been previously saved, the **Save As** dialog box opens.



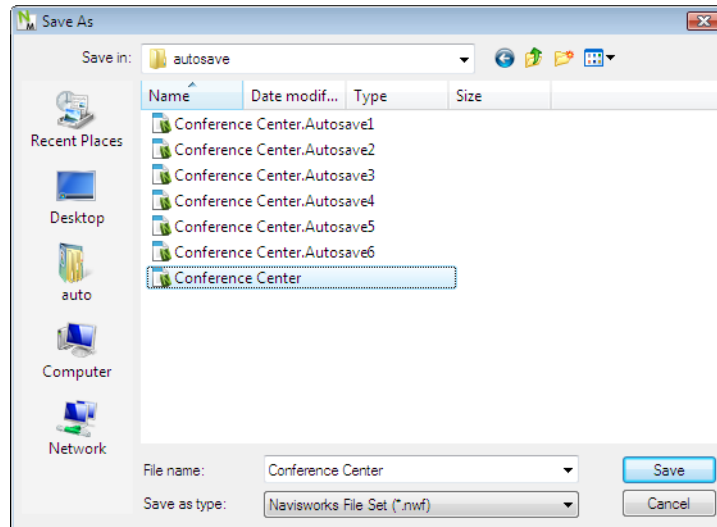
Enter the file name, use the **Files of Type** box to select one of the native Autodesk Navisworks formats (NWF or NWD), select the location for the file, and click **Save**.

TIP If you need to open your file with an earlier version of Autodesk Navisworks, save it as that version type.

 **Toolbar:** Classic user interface: **Standard** ► **Save** 
 **Command entry:** CTRL + S

To rename a file

- 1 Click the application button  ► **Save As** .
- 2 In the **Save As** dialog box, enter the new name for your file.



- 3 Click **Save**.

To publish an NWD file

- 1 Open the file that you want to publish (NWD or NWF).
- 2 Click the application button  **> Publish**  .
- 3 Use the **Publish** dialog box to enter the document information, and specify the required document protection. The text boxes in this dialog box remember history of up to five last entries. Clicking the down arrow on the far right side of the text box lets you select an entry instead of re-typing it.

Publish

Title: SHU Library

Subject: Extension of the main SHU Library

Author: Galyna Key

Publisher: Galina Key

Published For: SHU

Copyright: BCDesign

Keywords: library extension

Comments: Issue 4, Review only

Password:

☐ Display at password

☒ Expires: 20/01/2010

☐ May be re-saved

☐ Display on open

☒ Embed Textures

☐ Embed Database Properties

☐ Prevent Object Property Export

OK Cancel

- 4 Click **OK**.
- 5 In the **Save As** dialog box, enter the file name, and specify the file location.
- 6 Click **Save**. Your file is now published.

Ribbon: Output tab ► Publish panel ► NWD 

 **Toolbar: Classic user interface: Standard ► Publish** 

2D and Multi-Sheet Files

You can now work with 2D files and files containing multiple sheets/models.

Whilst multi-disciplinary models provide a real-world likeness of what a finished project should look like, various project stakeholders and field workers are more familiar with 2D plans and elevations. Autodesk Navisworks Manage

2012 supports 2D and multi-sheet files that can be reviewed, or combined with models to provide multiple representations of project data.

The supported 2D and multi-sheet file formats are: DWF, DWF(x), and native file formats (NWD and NWF).

When you open a supported file, which contains multiple sheets/models, the default sheet/model is displayed in the **Scene View**, and all of the file's sheets/models are listed in the **Project Browser** window. If a file contains both 3D models and 2D sheets, the 3D model is loaded and displayed in the **Scene View** by default. If you do not require 2D capabilities, simply close the **Project Browser** window and continue working in a 3D workspace.

Add Sheets/Models to the Currently Opened File

You can add 2D sheets and 3D models from DWF, DWF(x), NWD, or NWF files to the file currently open in Autodesk Navisworks.

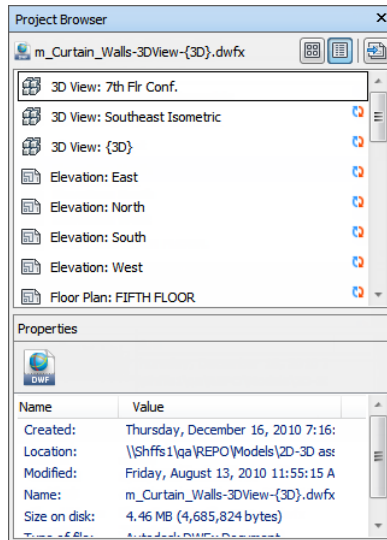
You can choose a source file in a DWF, DWF(x), NWD, or NWF format, add all of its sheets/models into the currently opened file, [delete](#) (page 216) any unwanted sheets/models, and then save the multi-sheet collection in a native NWD or NWF file format.

To add sheets/models to the currently opened file

- 1 Click the application button  **> Import > Sheets and Models** .
- 2 In the **Insert From File** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to the folder where your file is located.
- 3 Select the file, and click **Open**. All of the sheets/models in the selected files are added to the list in the Project Browser window, in the order they were listed in the original file.

Project Browser Window

The **Project Browser** is a dockable window, which lists all sheets/models in the currently opened file.



The Sheets/Models palette lists all sheets and models in the multi-sheet file. The label at the top of the palette indicates the file currently open in Autodesk Navisworks. The sheets/models can be represented as a list view or a thumbnail view. By default, the display order is the same as in the original file. The currently selected model/sheet is indicated with a shaded background, and the model/sheet currently opened in the **Scene View** is indicated with a black graphic border. When you open a multi-sheet file, not all sheets/models may have been prepared to be used in Autodesk Navisworks. The sheets/models which require preparation are indicated with the Prepare icon.

You can select several sheets/models at the same time with the SHIFT and CTRL keys, but you cannot open more than one sheet/model in the **Scene View**.

The **Properties** palette is used to examine properties for the sheet/model selected in the Sheets/Models palette. You can also view the properties for the currently opened file by clicking on its name. The properties are grouped by category, are read-only, and can be expanded/collapsed.

NOTE If more than one sheet/model is selected, the **Properties** palette only shows the number of selected items, and doesn't show any property information.

To toggle the Project Browser window

- Click **View** tab ► **Workspace** panel ► **Windows** drop-down, and select or clear the **Project Browser** check box.

Pointing device: Status bar ► **Project Browser** 

To view sheets/models as a list

- Click the **List View**  button in the top-right corner of the **Project Browser** window.

To view sheets/models as thumbnails

- Click the **Thumbnail View**  button in the top-right corner of the **Project Browser** window.

To examine sheet/model properties

- 1 Open a multi-sheet file.
- 2 Click the 2D sheet or 3D model in the **Project Browser**. The **Properties** palette shows the available information.

NOTE If more than one sheet/model is selected, the **Properties** palette only shows the number of selected sheets/models, and doesn't show any property information.

To open a sheet or model in the Scene View

- 1 Open the **Project Browser** window.
- 2 Double-click the sheet or model that you want to open.

TIP You can also use the multi-sheet navigation controls on the Status bar to quickly open sheets/models.

To rename a sheet/model

- 1 Open the **Project Browser** window.
- 2 Right-click the desired sheet or model, and click **Rename**.
- 3 Type a new name for your sheet or model, and press ENTER.

 **Command entry:** F2

To delete a sheet/model

- 1 Open the **Project Browser** window.
- 2 Right-click the sheet or model you want to remove, and click **Delete**. If you make a mistake, you can use the **Undo** command.

To prepare a sheet/model for use

- 1 Open the **Project Browser** window.
- 2 Right-click the sheet or model you want to prepare, and click **Prepare Sheet/Model** on the shortcut menu.

To prepare all sheets/models for use

- 1 Open the **Project Browser** window.
- 2 Right-click the sheet or model you want to prepare, and click **Prepare All Sheets/Models** on the shortcut menu.

To print a sheet/model

NOTE You can only print the current viewpoint.

- 1 Open the **Project Browser** window.
- 2 Double-click the sheet or model you want to print, to make it current (that is to open it in the **Scene View**).
- 3 Click **Output** tab ► **Print** panel ► **Print** .
- 4 Check the printer settings are as required, and click **OK**.

NOTE The maximum image size is 2048 x 2048 pixels.

The Properties button controls printer-specific ink and paper settings.

Sheets/Models Palette Shortcut Menu

Open Opens the selected sheet/model in the **Scene View**.

Delete Deletes the selected sheet/model in the list. You cannot delete the currently opened sheet/model.

Merge into Current Sheet Merges the 2D sheet selected in the list to the sheet currently opened in the **Scene View**. It is possible to merge the sheet onto itself.

Merge into Current Model Merges the 3D model selected in the list to the model currently opened in the **Scene View**. It is possible to merge the model onto itself.

Append to Current Sheet Appends the 2D sheet selected in the list to the sheet currently opened in the **Scene View**. It is possible to append the sheet onto itself.

Append to Current Model Appends the 3D model selected in the list to the model currently opened in the **Scene View**. It is possible to append the model onto itself.

Prepare Sheet/Model Prepares all sheets/models selected in the list for use in Autodesk Navisworks. To select multiple sheets/models, use SHIFT and CTRL keys. If the selected sheets/models are ready, this option is not available.

Prepare All Sheets/Models Prepares all sheets/models in the multi-sheet file for use in Autodesk Navisworks. If the sheets/models are ready for use, this option is not available.

Print Prints the sheet/model currently opened in the **Scene View**.

Rename Renames the sheet/model selected in the list. By default, the sheet/model names are the same as in the source file.

Buttons

Thumbnail View  Displays sheets/models as the thumbnail images in the Sheets/Models palette.

List View  Displays sheets/models as the list items in the Sheets/Models palette.

Import Sheets & Models  Adds sheets/models from the external files to the currently opened file in Autodesk Navisworks. All of the sheets/models in the selected files are added to the list in the Sheets/Models palette, in the order they were listed in the original file.

Work with 2D and Multi-Sheet Files

Each sheet and model in the **Project Browser** has its own self-contained representation within Autodesk Navisworks, enabling you to review and work with each sheet/model as though it were a separate file.

Each sheet and model has its own hierarchy of content, represented in the **Selection Tree**, which automatically updates to reflect the sheet/model currently open in the **Scene View**.

When reviewing sheets/models from the **Project Browser** you can:

- [Search for](#) (page 349) and [select](#) (page 341) objects within the current scene view.

Items on a 2D sheet remain intelligent objects, enabling you to select them for review or manipulation.

- Review object [properties](#) (page 367).

Items in a 2D DWF file exported from Autodesk Revit software for example, will contain similar properties to the corresponding 3D model. This enables searching for items based on properties and is also essential for 2D-3D association and the linking of items between these representations. See [Find All Sheets and Models Containing the Selected Object](#) (page 356).
- Apply [transforms and override object appearances](#) (page 382).

Selected objects can be manipulated, for example you can change their color or position.
- Add [links](#) (page 427).

You can add links that point to various data sources, such as manufacturer details, equipment manuals or specifications.
- Save [viewpoints](#) (page 442).

You can create and manage different views of your sheets (or models) so that you can jump to preset viewpoints without having to navigate each time to reach an item.
- Add [review markup](#) (page 403).

You can mark up viewpoints with redline annotations, or add comments.
- Take [measurements](#) (page 391) and capture these as [interactive dimensions](#) (page 399).

Measurements can be taken between points on a sheet/model. Object snapping is also enabled making it easier to take accurate measurements.

Add Geometry and Metadata to the Current Sheet/Model

You can append and merge individual sheets/models within the same multi-sheet file. Any duplicate geometry or markup is removed when sheets or models are merged.

For multi-sheet files, you can bring geometry and data from internal project sources, that is 2D sheets or 3D models listed in the **Project Browser**, into the currently open sheet or model. However, 3D models can only be merged/appended with other 3D models, and 2D models can only be merged/appended with other 2D sheets.

To append 2D sheets in a multi-sheet file

- 1 Open a multi-sheet file.
- 2 If the **Project Browser** window is not displayed, click  on the Status bar.
- 3 Double-click the desired 2D sheet in the **Project Browser** to open it in the **Scene View**.
- 4 Use the **Project Browser** to select all 2D sheets that you want to append to the currently open sheet.

TIP To select multiple sheets, use SHIFT and CTRL keys.

- 5 Right-click the selection, and click **Append to Current Sheet**.

NOTE The **Undo** command is not available.

To append 3D models in a multi-sheet file

- 1 Open a multi-sheet file.
- 2 If the **Project Browser** window is not displayed, click  on the Status bar.
- 3 Double-click the desired 3D model in the **Project Browser** to open it in the **Scene View**.
- 4 Use the **Project Browser** to select all 3D models that you want to append to the currently open model.

TIP To select multiple model, use SHIFT and CTRL keys.

- 5 Right-click the selection, and click **Append to Current Model**.

NOTE The **Undo** command is not available.

To merge 2D sheets in a multi-sheet file

- 1 Open a multi-sheet file.
- 2 If the **Project Browser** window is not displayed, click  on the Status bar.
- 3 Double-click the desired 2D sheet in the **Project Browser** to open it in the **Scene View**.
- 4 Use the **Project Browser** to select all 2D sheets that you want to merge to the currently open sheet.

TIP To select multiple sheets, use SHIFT and CTRL keys.

- 5 Right-click the selection, and click **Merge to Current Sheet**. Any duplicate geometry and markup is automatically removed.

NOTE The **Undo** command is not available.

To merge 3D models in a multi-sheet file

- 1 Open a multi-sheet file.
- 2 If the **Project Browser** window is not displayed, click  on the Status bar.
- 3 Double-click the desired 3D model in the **Project Browser** to open it in the **Scene View**.
- 4 Use the **Project Browser** to select all 3D models that you want to merge to the currently open model.

TIP To select multiple model, use SHIFT and CTRL keys.

- 5 Right-click the selection, and click **Merge to Current Model**.

NOTE The **Undo** command is not available.

Complex Datasets

You can use Autodesk Navisworks to combine design files together into complex datasets.

Autodesk Navisworks enables you to combine design files together by bringing geometry and metadata from supported external files into the current scene. Autodesk Navisworks automatically aligns rotation and origin of models, and rescales the units in each appended file to match [display units](#) (page 157). If the rotation, origin, or units are incorrect for the scene, you can manually adjust them for each of the incorporated files.

For multi-sheet files, you can also bring geometry and data from internal project sources, that is 2D sheets or 3D models listed in the **Project Browser**, into the currently open sheet or model. See [Add Geometry and Metadata to the Current Sheet/Model](#) (page 219).

Append Geometry and Metadata to the Current Scene

You can append geometry and data from the selected files to the current 3D model or 2D sheet.

NOTE If you try to drag and drop files directly into the **Selection Tree** window or to use the **Open** dialog box, you will create a new Autodesk Navisworks file instead.

To append files

- 1 Click **Home** tab ► **Project** panel ► **Append** .
- 2 In the **Append** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to the folder where the files you want to add is located.
- 3 Select the required files, and click **Open**.

TIP To select multiple files, use SHIFT and CTRL keys.

 **Command entry:** CTRL + A

 **Toolbar:** Classic user interface: **Standard** ► **Append** 

Delete Files

You can delete files appended to your Autodesk Navisworks file.

NOTE You cannot delete any files from a published or saved NWD file. To delete files use the NWF file format.

In multi-sheet files, you can [delete](#) (page 216) any unwanted sheets/models.

To delete a file

- 1 Open a Autodesk Navisworks file.
- 2 Right-click the file you want to delete in the **Selection Tree**.
- 3 Click **Delete**.
- 4 Click **Yes** to confirm your command.

NOTE The **Undo** command is not available; therefore, if you make a mistake, you will need to append your file back manually.

 **Menu:** Classic user interface: **File > Delete**

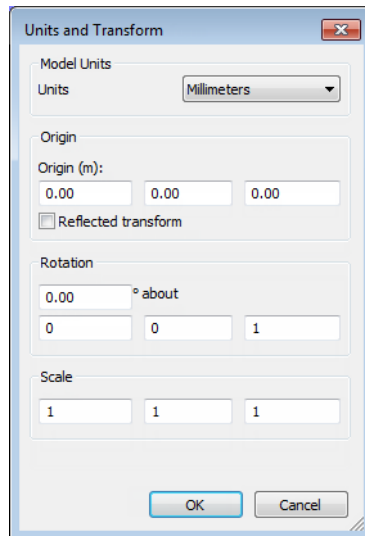
Adjust Units and Transform

You can change the units, and modify the rotation, origin, and scale of each appended 3D model or 2D sheet.

It is recommended to adjust units first, before attempting to fix the problems with model or sheet alignment.

To change units in a loaded 3D file

- 1 Right-click the desired 3D file in the **Selection Tree**, and click **Units and Transform** on the shortcut menu.
- 2 In the **Units and Transform** dialog box, select the required format in the **Units** drop-down list.



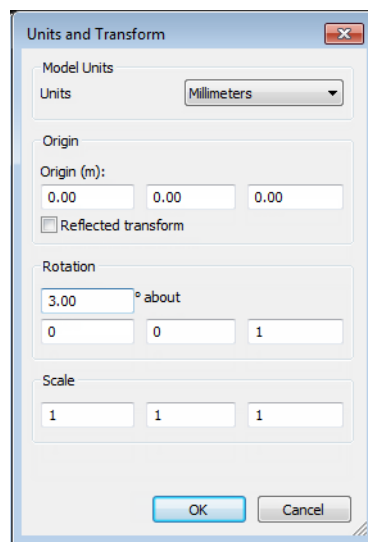
- 3 Click **OK**.

To change units in a loaded 2D sheet

- 1 Right-click the desired 2D sheet in the **Scene View**, and click **Units and Transform** on the shortcut menu.
- 2 In the **Units and Transform** dialog box, select the required format in the **Units** drop-down list.
- 3 Click **OK**.

To change a transform in a loaded 3D file

- 1 Right-click the desired 3D file in the **Selection Tree**, and click **Units and Transform** on the shortcut menu.
- 2 To move the model's location, in the **Units and Transform** dialog box, the **Origin** area, enter values for the X, Y and Z axis.
If you use a negative scale, select the **Reflected Transform** check box.
- 3 To change the model's rotation, in the **Units and Transform** dialog box, the **Rotation** area, enter an angle of rotation, and selecting the axis to rotate about (by typing a value greater than 0). This will rotate the model about its origin point.
- 4 To change the model's scale, in the **Units and Transform** dialog box, the **Scale** area, enter values for the X, Y and Z axis. To proportionally scale an object, ensure the X, Y, and Z values are equal.
Entering a negative value flips the model inside out.



- 5 Click **OK**.

To change a transform in a loaded 2D sheet

- 1 Right-click the desired 2D sheet in the **Scene View**, and click **Units and Transform** on the shortcut menu.
- 2 To move the sheet's location, in the **Units and Transform** dialog box, the **Origin** area, enter values for the X and Y axis.
- 3 To change the sheet's rotation, in the **Units and Transform** dialog box, the **Rotation** area, enter an angle of rotation. This will rotate the sheet about its origin point.
- 4 To change the sheet's scale, in the **Units and Transform** dialog box, the **Scale** area, enter values for the X and Y axis. To proportionally scale a sheet, ensure the X and Y values are equal.
- 5 Click **OK**.

Refresh Files

When working in Autodesk Navisworks, it is possible that others may be working on the CAD files you are currently reviewing. For example, if you are coordinating various disciplines on a project, then you may have an NWF file referencing numerous design files. During the iterative stages of the project, any member of the design team could potentially be modifying their CAD files.

To ensure the data you are reviewing is current, Autodesk Navisworks provides a refresh function to reopen the files that have been modified on the disk since commencing the review session.

To refresh currently open files

- 1 Click **Refresh**  on the **Quick Access** toolbar.

 **Toolbar:** Classic user interface: **Standard** ► **Refresh** 

Merge Files

Autodesk Navisworks is a collaborative solution, and although users may be reviewing the model in different ways, their resultant files can be merged into

a single Autodesk Navisworks file, with any duplicate geometry and markup automatically removed.

When merging multiple NWF files that each comprise the same referenced files, Autodesk Navisworks only loads a single set of the combined models, along with all review markup (such as tags, viewpoints or comments) from each NWF file. Any duplicate geometry or markup is removed when merged.

For multi-sheet files, you can also merge geometry and data from internal project sources, that is 2D sheets or 3D models listed in the **Project Browser**, into the currently open sheet or model. See [Add Geometry and Metadata to the Current Sheet/Model](#) (page 219).

Merging TimeLiner Data

For files that contain TimeLiner data, merging is done as follows:

- if two TimeLiner data sets are identical, all duplicate data is removed.
- if two TimeLiner data sets contain the same Primary link, then the data set with the most recent link (that is, the newest file date) is chosen over the other. If the Primary links are different, then the data set with the highest number of tasks will be used, and links are re-attached wherever possible.
- if two TimeLiner data sets are completely unrelated, no data merging is performed.

To merge files

- 1 Click **New**  on the **Quick Access** toolbar.
- 2 Open the first of the files with the review markup.
- 3 Click **Home** tab ► **Project** panel ► **Merge** .
- 4 In the **Merge** dialog box, use the **Files of Type** box to select the appropriate file type (NWD or NWF), and navigate to the folder where your files you want to merge are located.
- 5 Select the required files, and click **Open**.

TIP To select multiple files, use SHIFT and CTRL keys.

 **Command entry:** CTRL + M

 **Toolbar:** Classic user interface: **Standard** ► **Merge** 

Email Files

You can send and receive Autodesk Navisworks files by email.

To send an open Autodesk Navisworks file, click the application button 

► **Send by Email**  .

Autodesk Navisworks saves your file first, and then opens the available mail software and attaches your file to a blank email message.

You can email both NWD and NWF files, although the size of an NWD file will be considerably larger.

Ribbon: Output tab ► Send panel ► **Send by Email** 

 **Toolbar: Standard** ► **Send by Email** 

Receive Files

To open a received file, save the attachment on your disk, and then double-click it.

If it's an NWF file, Autodesk Navisworks searches for the referenced files first using the absolute path that the sender originally saved the file with. This is useful if a team is on a local network and the files can be found using the Universal Naming Convention (UNC).

TIP A team not sharing a server can organize a project using the same file hierarchy and drive letter, and Autodesk Navisworks can find the files this way.

If Autodesk Navisworks is unable to find the referenced files, then you need to save the attached NWF file in a directory where all referenced files are located. Autodesk Navisworks can then search for the files relative to the master NWF location.

TIP You can use the same method to move a whole sub-directory from your projects directory to a completely new location. Save the master NWF file in this new place, and Autodesk Navisworks will be able to search for the referenced files from there.

Batch Utility

You can now use the Batch Utility to automate common file importing/conversion processes.

The Batch Utility is integrated with Windows Task Scheduler to allow you to set up tasks to be run automatically at set times and intervals.

Use Batch Utility

You can use the Batch Utility to run and schedule the following commands:

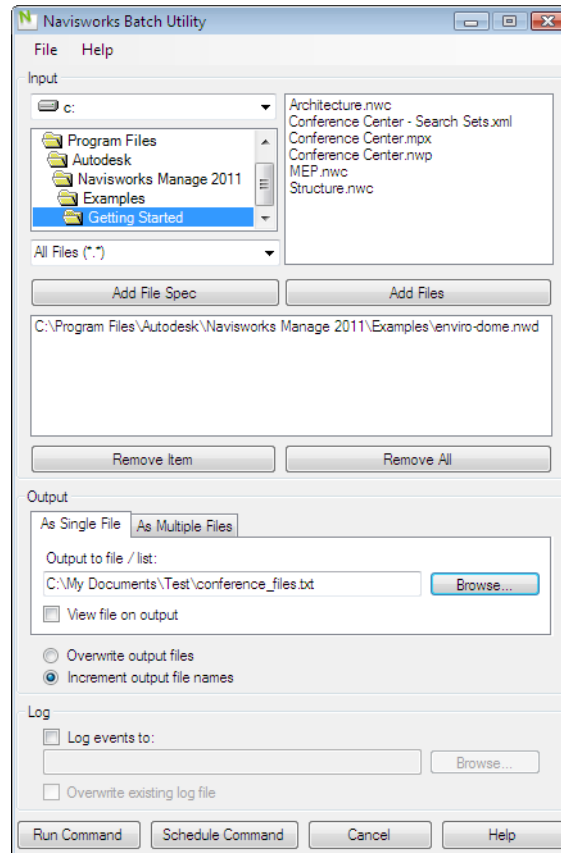
- Create a list of all design files in the currently open Autodesk Navisworks file.
- Append multiple design files into a single NWD or an NWF file.
- Convert multiple design files into individual NWD files.

IMPORTANT You can use any of the currently supported file formats when you append or convert files with the Batch Utility. However, if you use any published NWD files as input, they must have been created with the **May Be Resaved** option selected, otherwise when you run the **Batch Utility**, no NWD output files can be saved. It is still possible to create an NWF output file.

All events can be recorded in a log file.

To generate a list of all design files used in the current model

- 1 In Autodesk Navisworks application, open the desired Autodesk Navisworks file, and click **Home** tab ► **Tools** panel ► **Batch Utility** .
- 2 The Autodesk Navisworks **Batch Utility** dialog box is displayed, and the path to the current model is automatically added to the **Input** area of the dialog box.
- 3 In the **Output** area, **As Single File** tab, click the **Browse** button.
- 4 In the **Save Output As** dialog box, browse to the desired folder, and enter the name for the text file.
- 5 In the **Save as Type** drop-down list, select the **File List (*.txt)** option, and click **Save**.



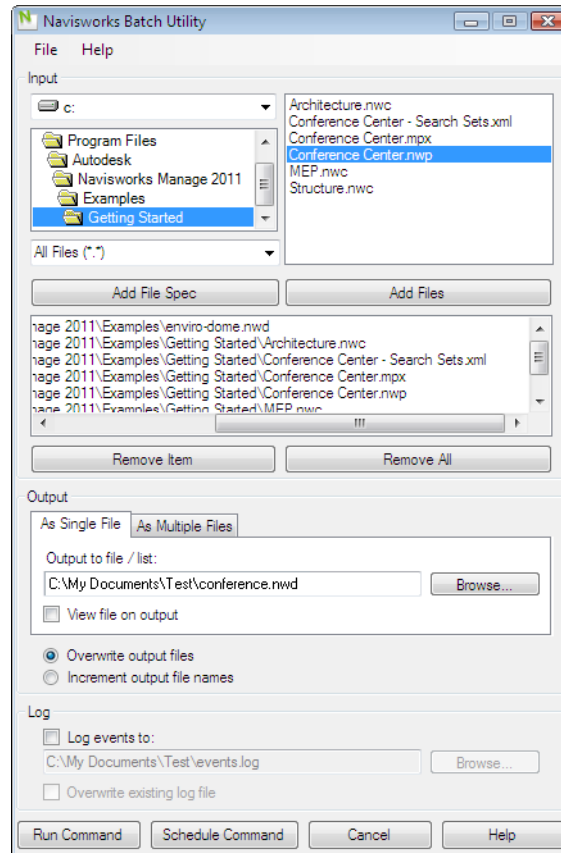
- 6 In the Autodesk Navisworks **Batch Utility** dialog box, click the **Run Command** button.

TIP If you want to automatically open the file as soon as it's been created, select the **View File on Output** check box before you click the **Run Command** button.

To append multiple design files into a single Autodesk Navisworks file

- 1 In Autodesk Navisworks application, click **Home** tab ► **Tools** panel ► **Batch Utility** .

- 2 In the Autodesk Navisworks **Batch Utility** dialog box, use the **Input** area to create a list of design files for conversion:
 - Select the drive and the folder where the design files are located. Once the desired location is selected, use the File Type menu below to filter the design files that are displayed on the right. The menu contains a list of all currently supported file formats.
 - There are two methods you can use to choose the files for conversion. You can double-click files to add them to the conversion list.
Alternatively, you can click the **Add File Spec** button. This will make Autodesk Navisworks convert the files located in the selected folder. The option chosen in the File Type menu determines which files are converted.
- 3 In the **Output** area, **As Single File** tab, click the **Browse** button.
- 4 In the **Save Output As** dialog box, browse to the desired folder, and enter the name for the new file.
- 5 In the **Save as Type** drop-down list, select the required file format (NWD or NWF), and click **Save**.
- 6 To automatically open the file as soon as it's been created, select the **View File on Output** check box.
- 7 Select the **Increment Output File Names** radio button if you want to append a four digit number to the end of file name. By default, the old output files are overwritten.



8 Click the **Run Command** button.

To convert multiple design files into individual NWD files

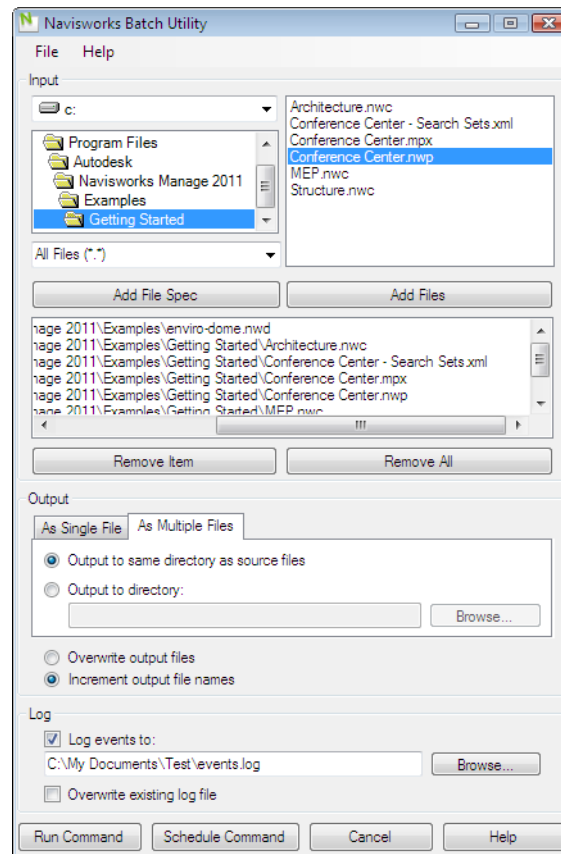
- 1 In Autodesk Navisworks application, click **Home** tab ► **Tools** panel ► **Batch Utility** .
- 2 In the Autodesk Navisworks **Batch Utility** dialog box, use the **Input** area to create a list of design files for conversion:
 - Select the drive and the folder where the design files are located. Once the desired location is selected, use the File Type menu below to filter the design files that are displayed on the right. The menu contains a list of all currently supported file formats.
 - There are two methods you can use to choose the files for conversion. You can double-click files to add them to the conversion list.

Alternatively, you can click the **Add File Spec** button. This will make Autodesk Navisworks convert the files located in the selected folder. The option chosen in the File Type menu determines which files are converted.

- 3 In the **Output** area, **As Multiple Files** tab, choose the location for the converted files. By default, files are created in the same location as source files.

If you want to change the output location, select the **Output to Directory** radio button, and click the **Browse** button. Use the **Browse for Folder** dialog box to select the desired folder.

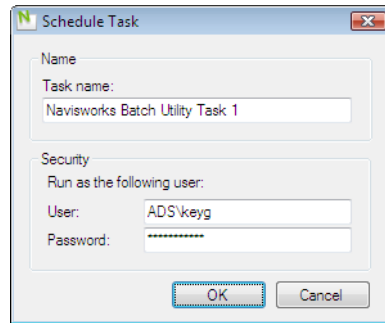
- 4 Select the **Increment Output File Names** radio button if you want to append a four digit number to the end of file name. By default, the old output files are overwritten.



- 5 Click the **Run Command** button.

To schedule appending multiple design files into a single Autodesk Navisworks file

- 1 In Autodesk Navisworks application, click **Home** tab ► **Tools** panel ► **Batch Utility** .
- 2 In the Autodesk Navisworks **Batch Utility** dialog box, use the **Input** area to create a list of design files for conversion:
 - Select the drive and the folder where the design files are located. Once the desired location is selected, use the File Type menu below to filter the design files that are displayed on the right. The menu contains a list of all currently supported file formats.
 - There are two methods you can use to choose the files for conversion. You can double-click files to add them to the conversion list.
Alternatively, you can click the **Add File Spec** button. This will make Autodesk Navisworks convert the files located in the selected folder. The option chosen in the File Type menu determines which files are converted.
- 3 In the **Output** area, **As Single File** tab, click the **Browse** button.
- 4 In the **Save Output As** dialog box, browse to the desired folder, and enter the name for the new file.
- 5 In the **Save as Type** drop-down list, select the required file format (NWD or NWF), and click **Save**.
- 6 Select the **Increment Output File Names** radio button to append a four digit number to the end of file name. By default, the old output files are overwritten.
- 7 Click the **Schedule Command** button.
- 8 In the **Save Task File As** dialog box browse to the desired location, and click the **Save** button.
- 9 In the **Schedule Task** dialog box, change the task name, if required, and enter your user name and password. Click **OK**.



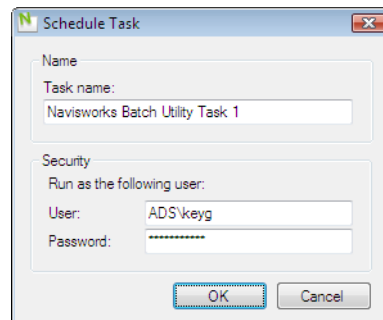
NOTE Task Scheduling does not work for Windows XP/2000 users running local accounts (that is, not on a domain) without passwords.

- 10 In the **Windows Task Scheduler**, the **Schedule** tab, click the **New** button, and specify when and how often the task is to run. You can add as many schedules as necessary.
- 11 Click **OK**.

To schedule conversion of multiple design files into individual NWD files

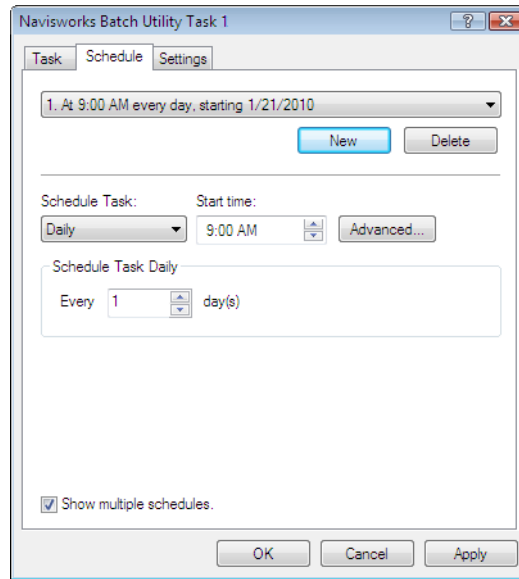
- 1 In Autodesk Navisworks application, click **Home** tab ► **Tools** panel ► **Batch Utility** .
- 2 In the Autodesk Navisworks **Batch Utility** dialog box, use the **Input** area to create a list of design files for conversion:
 - Select the drive and the folder where the design files are located. Once the desired location is selected, use the File Type menu below to filter the design files that are displayed on the right. The menu contains a list of all currently supported file formats.
 - There are two methods you can use to choose the files for conversion. You can double-click files to add them to the conversion list. Alternatively, you can click the **Add File Spec** button. This will make Autodesk Navisworks convert the files located in the selected folder. The option chosen in the File Type menu determines which files are converted.
- 3 In the **Output** area, **As Multiple Files** tab, choose the location for the converted files. By default, files are created in the same location as source files. If you want to change the output location, select the **Output to Directory** radio button, and click the **Browse** button. Use the **Browse for Folder** dialog box to select the desired folder.

- 4 Select the **Increment Output File Names** radio button to append a four digit number to the end of file name. By default, the old output files are overwritten.
- 5 Click the **Schedule Command** button.
- 6 In the **Save Task File As** dialog box browse to the desired location, and click the **Save** button.
- 7 In the **Schedule Task** dialog box, change the task name, if required, and enter your user name and password. Click **OK**.



NOTE Task Scheduling does not work for Windows XP/2000 users running local accounts (that is, not on a domain) without passwords.

- 8 In the **Windows Task Scheduler**, the **Schedule** tab, click the **New** button, and specify when and how often the task is to run. You can add as many schedules as necessary.



9 Click **OK**.

To create a log file

- 1 In Autodesk Navisworks application, click **Home** tab ► **Tools** panel ► **Batch Utility** .
- 2 In the **Log** area, select the **Log Events To** check box.
- 3 Click the **Browse** button.
- 4 In the **Save Log As** dialog box, browse to the desired folder, enter the name for your log, and click **Save**.
- 5 Clear the **Overwrite Existing Log File** check box if you want to keep new and old events in the same log file. The new information is appended to the bottom of the log file.

Command Line Options for Batch Utility

You can use command line switches to covert files with the Batch Utility. To do this, you need to create a list of files that you want to process. This file

must be a UTF-8 encoded text file containing file paths or file specs, one on each line, and without any quotation marks. For example:

Command Line Switch	Description
/i	Indicates the next argument is the input file. You must provide the file name and file path within quotation marks.
/of	Outputs to a single Navisworks file (NWD or NWF). The file name and file path should be provided within quotation marks. For example, "C:\temp\a.nwd" or "C:\temp\a.nwf".
/od	Outputs multiple NWD files into a specific folder. The folder path should be provided within quotation marks. For example, "C:\temp".
/osd	Outputs multiple NWD files into the same folder as the input files.
/view	Automatically open the file as soon as it's been created.
/over	Overwrites output files. This is the default behavior.
/inc	Increments output file names.
/log	Outputs events into a log file. You must provide the file name and file path within quotation marks. For example, "C:\temp\log.log".
/appndlog	Records new and old events in the same log file. The new information is appended to the bottom of the log file.

Command Line Switch	Description
/lang	Specifies the language to use when logging. You can use one of the following arguments: en-US = English, de-DE = German, es-ES = Spanish, fr-FR = French, it-IT = Italian, ja-JP = Japanese, ko-KR = Korean, pt-BR = Brazilian Portuguese, ru-RU = Russian, zh-CN = Chinese (PRC).

The syntax for using command line switches is:

```
FileToolsTaskRunner.exe /i <input file> [/of <output file>/od <folder>/osd] [/view]
[/overl/inc] [/log <log file>] [/append] [/lang <language>]
```

For example, the following entry processes the list of design files in the *filetools.txt* file, appends them into a single *conference.nwd* file, and creates a log file *events.log* in *French* language.

```
"C:\Program Files\Autodesk\Navisworks Manage
2012\FileToolsTaskRunner.exe" /i "C:\Conference Center\filetools.txt"
/of "C:\Conference Center\conference.nwd" /log "C:\Conference
Center\events.log" /lang fr-fr
```

View Scene Statistics

Scene statistics display information about the current scene.

Scene statistics list all files contributing to the scene, and the different graphic elements that make up the scene, along with which of these have been processed or ignored when loaded.

Other useful statistics are the bounding box of the entire scene and the total number of primitives (triangles, lines, points) in the scene.

To view scene statistics

- 1 Click the **Home** tab and slide out the **Project** panel.
- 2 Click **Scene Statistics** .

Explore Your Model

5

Navigate a Scene

In Autodesk Navisworks, you have a variety of options for navigating your scene.

You can directly manipulate your position in the **Scene View** with navigation tools on the navigation bar, such as the pan and zoom. You can also use **SteeringWheels®** that travel with the cursor, and can save you time by combining many of the common navigation tools into a single interface.

You can use the **ViewCube®**, a 3D navigational tool that enables you to reorient the view of a model by clicking predefined areas on the cube. For example, clicking the front of the ViewCube turns the view until the camera is facing the front of the scene. You can also click the ViewCube and drag it to rotate the view freely. The ViewCube is not available in a 2D workspace.

TIP Use a 3Dconnexion 3D mouse as an alternative to the mouse to navigate and change the orientation of your model in a 3D workspace. There is a 2D mode you can use to navigate in a 2D workspace.

When you navigate a 3D model, you can use the realism tools on the **Viewpoint** tab ► **Navigate** panel to control the speed and realism of your navigation. So, for example, you can walk down stairs or follow terrain, crouch under objects, and use an avatar to represent yourself within the 3D model. The realism tools are not available in a 2D workspace.

Animating Navigation

As you navigate in Autodesk Navisworks, you can record viewpoint animations, and then play them back. For more information, see [Play Back Animations](#) (page 471).

Orientation in a 3D Workspace

Although Autodesk Navisworks uses the X, Y, Z coordinate system, there is no hard-and-fast rule as to which way each of these particular axes actually “points”.

Autodesk Navisworks reads the data necessary to map which way is “up” and which way is “north” directly from the files loaded into your scene. If this is not possible, by default, Z is treated as “up” and Y is treated as “north”.

It is possible to change the “up” and “north” directions for the entire model (world orientation), and the “up” direction for the current viewpoint (viewpoint up vector).

NOTE Changing the viewpoint up vector affects navigation in modes that rely on the “up” direction of the current viewpoint, such as **Walk**, **Constrained Orbit**, and **Orbit**. It also has an impact on section views.

To align the viewpoint up vector to the current view

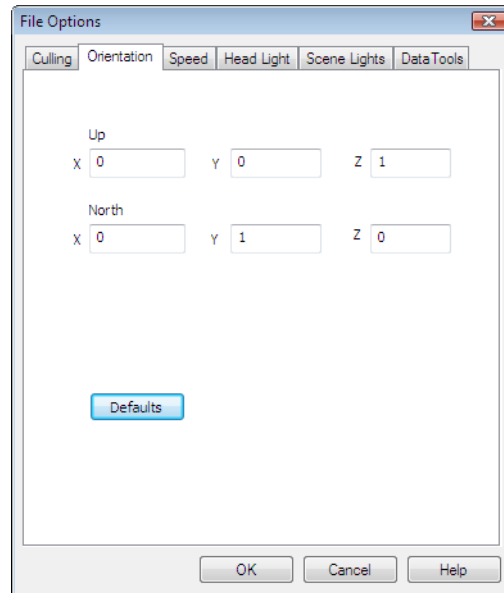
- In **Scene View**, right-click and click **Viewpoint ► Set Viewpoint Up ► Set Up** on the shortcut menu.

To align the viewpoint up vector to one of the preset axes

- 1 In **Scene View**, right-click and click **Viewpoint ► Set Viewpoint Up**.
- 2 Click one of the preset axis. Choose from:
 - **Set Up + X**
 - **Set Up - X**
 - **Set Up + Y**
 - **Set Up - Y**
 - **Set Up + Z**
 - **Set Up - Z**

To change the world orientation

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **Orientation** tab, enter the required values to adjust the model orientation.



3 Click **OK**.

Product-Specific Navigation Tools

The navigation bar and SteeringWheels provide you with access to a set of product-specific navigation tools in 2D and 3D workspaces.

In Autodesk Navisworks Manage 2012, some classic navigation tools have been replaced with new navigation tools. See the table below for more details.

Classic tool	Replaced by	2D?	Can be switched back to classic?
Pan	Pan	Yes	No
Zoom	Zoom	Yes	No
View All	Zoom All	Yes	No
View Selected	Zoom Selected	Yes	No

Classic tool	Replaced by	2D?	Can be switched back to classic?
Zoom Box	Zoom Window	Yes	No
Orbit	Orbit	No	Yes
Examine	Free Orbit	No	Yes
Turntable	Constrained Orbit	No	Yes
Look Around	Look	No	No
Focus	-	-	-
Walk	Walk	No	Yes
Fly	-	No	-

Customizing Tools Behavior

For the navigation bar, you can use **Options Editor** to toggle between standard and classic modes for **Orbit** and **Walk** tools.

Control Navigation Realism

You can use [Collision](#) (page 312), [Gravity](#) (page 311), [Crouch](#) (page 311), and [Third Person View](#) (page 313) to enhance your navigation experience in a 3D workspace.

TIP Use a combination of **Collision**, **Gravity**, and **Crouch** with the **Walk** tool. This allows you, for example, to walk up and down stairs and walk under low objects.

Navigation Bar Tools

The navigation bar contains a set of product-specific navigation tools.

Pan Tool

The pan tool moves the view parallel to the screen.

The tool is activated by clicking **Pan**  on the navigation bar. **Pan** behaves the same way as the pan tool available on the SteeringWheels.

Zoom Tools

Set of navigation tools for increasing or decreasing the magnification of the current view of the model.

The following zoom tools are available:

- **Zoom Window**  . Allows you to draw a box and zoom into that area.
- **Zoom**  . Standard click/drag zoom.
- **Zoom Selected**  . Zooms in/out to show the selected geometry.
- **Zoom All**  . Zooms out to show the whole scene.

Zoom Window

The tool is activated by clicking **Zoom Window** in the **Zoom** drop-down on the navigation bar. It behaves the same way as the classic **Zoom Box** mode.

Zoom

The tool is activated by clicking **Zoom** in the **Zoom** drop-down on the navigation bar. It behaves the same way as the **Zoom** tool available on the SteeringWheels.

Zoom Selected

The tool is activated by clicking **Zoom Selected** in the **Zoom** drop-down on the navigation bar. Alternatively, you can click **Item Tools** tab ► **Look At**

panel ► **Zoom**  on the ribbon. It behaves the same way as the classic **View Selected** tool.

Zoom All

The tool is activated by clicking **Zoom All** in the **Zoom** drop-down on the navigation bar. It behaves the same way as the classic **View All** tool.

Orbit Tools

Set of navigation tools for rotating the model around a pivot point while the view remains fixed. These tools are not available in a 2D workspace.

The following orbit tools are available:

- **Orbit**  . Moves the camera around the focal point of the model. The up direction is always maintained, and no camera rolling is possible.
- **Free Orbit**  . Rotates the model around the focal point in any direction.
- **Constrained Orbit**  . Spins the model around the up vector as though the model is sitting on a turntable. The up direction is always maintained.

Orbit

The tool is activated by clicking **Orbit** in the **Orbit** drop-down on the navigation bar. It behaves the same way as the **Orbit** tool on the SteeringWheels. You can use the **Options Editor** to switch back to the classic **Orbit** mode.

Free Orbit

The tool is activated by clicking **Free Orbit** in the **Orbit** drop-down on the navigation bar. It behaves similarly to the classic **Examine** mode. You can use the **Options Editor** to switch back to the classic **Examine** mode.

Constrained Orbit

The tool is activated by clicking **Constrained Orbit** in the **Orbit** drop-down on the navigation bar. It behaves similarly to the classic **Turntable** mode. You can use the **Options Editor** to switch back to the classic **Turntable** mode.

To use the classic Orbit tool with the navigation bar

- 1 On the navigation bar, click **Customize** ► **Navigation Bar Options**.

- 2 In the **Options Editor**, the **Navigation Bar** page under the **Interface** node, select the **Use Classic Orbit** check box in the **Orbit Tools** area.
- 3 Click **OK**.

To use the classic Examine tool with the navigation bar

- 1 On the navigation bar, click **Customize > Navigation Bar Options**.
- 2 In the **Options Editor**, the **Navigation Bar** page under the **Interface** node, select the **Use Classic Free Orbit (Examine)** check box in the **Orbit Tools** area.
- 3 Click **OK**.

To use the classic Turntable tool with the navigation bar

- 1 On the navigation bar, click **Customize > Navigation Bar Options**.
- 2 In the **Options Editor**, the **Navigation Bar** page under the **Interface** node, select the **Use Classic Constrained Orbit (Turntable)** check box in the **Orbit Tools** area.
- 3 Click **OK**.

Look Tools

Set of navigation tools for rotating the current view vertically and horizontally. These tools are not available in a 2D workspace.

The following look tools are available:

- **Look Around**  . Looks around the scene from the current camera location.
- **Look At**  . Looks at a particular point in the scene. The camera moves to align with that point.
- **Focus**  . Looks at a particular point in the scene. The camera stays where it is.

Look Around

The tool is activated by clicking **Look Around** in the **Look** drop-down on the navigation bar. It behaves the same way as the **Look** tool available on the SteeringWheels.

Look At

The tool is activated by clicking **Look At** in the **Look** drop-down on the navigation bar. It behaves the same way as the SteeringWheels **Look** tool when you press and hold the SHIFT key.

Focus

The tool is activated by clicking **Focus** in the **Look** drop-down on the navigation bar. See [Focus](#) (page 310) for more details.

Walk and Fly Tools

Set of navigation tools for moving around the model and controlling realism settings. These tools are not available in a 2D workspace.

The following tools are available:

- **Walk** . Moves through a model as if you were walking through it.
- **Fly** . Moves through a model like in a flight simulator.

Walk

The tool is activated by clicking **Walk** in the **Walk/Fly** drop-down on the navigation bar. By default, the tool behaves like the **Walk** tool on the SteeringWheels. You can customize the tool options in the **Options Editor**. You can also switch back to the classic **Walk** mode.

Fly

The tool is activated by clicking **Fly** in the **Walk/Fly** drop-down on the navigation bar. It behaves the same way as the classic **Fly** mode.

See also:

[Control the Realism of Your Navigation](#) (page 311)

To use classic Walk tool with the navigation bar

- 1 On the navigation bar, click **Customize** ► **Navigation Bar Options**.
- 2 In the **Options Editor**, the **Navigation Bar** page under the **Interface** node, select the **Use Classic Walk** check box in the **Walk Tool** area.

- 3 Click **OK**.

SteeringWheels Tools

Each wheel is divided into different wedges. Each wedge contains a navigation tool used to reorient the current view of a model. Which navigation tools are available depends on which wheel is active.

Center Tool

With the Center tool, you can define the center of the current view of a model. To define the center, drag the cursor over your model. A sphere (pivot point) is displayed in addition to the cursor. The sphere indicates that the point below the cursor in the model will be the center of the current view when you release the mouse button. The model is centered on the sphere.

NOTE If the cursor is not over the model, the center cannot be set and a prohibited cursor is displayed.



The point defined by the Center tool provides a focal point for the Zoom tool and a pivot point for the Orbit tool.

NOTE If you want to zoom from the Full Navigation wheels from your defined center point, hold down **CTRL** before zooming.

To specify a point on a model as the center of a view

- 1 Display one of the Full Navigation wheels or the big View Object wheel.
- 2 Click and hold down the Center wedge.

- 3 Drag the cursor to the desired location of the model.
- 4 Release the button on your pointing device when the sphere is displayed.
The model is panned until the sphere is centered.

To specify the target point for the Zoom and Orbit tools

- 1 Display one of the Full Navigation wheels or the big View Object wheel.
- 2 Click and hold down the Center wedge.
- 3 Drag the cursor over the desired location of the model.
- 4 Release the button on your pointing device when the sphere is displayed.
The model is panned until the sphere is centered.
- 5 Use the Zoom or Orbit tool to reorient the view of the model.
If you are using one of the Full Navigation wheels, hold down the **CTRL** key before using the Zoom tool.

Forward Tool

You use the Forward tool to change the magnification of the model by increasing or decreasing the distance between the current point of view and the pivot point. The distance that you can move forward or backward is limited by the position of the pivot point.



NOTE In orthographic views, the Forward tool is limited to the distance between the current position and the pivot point. In perspective views, it is not limited, so you can move the cursor through the pivot point.

To adjust the distance between the current point of view and the pivot point you use the Drag Distance indicator. The Drag Distance indicator has two marks on it that show the start and destination distances from the current point of view. The current traveled distance is shown by the orange position indicator. Slide the indicator forward or backwards to decrease or increase the distance towards the pivot point.

To reorient a view by moving towards or away from the model

- 1 Display the big Tour Building wheel.
- 2 Click and hold down the Forward wedge.
The Drag Distance indicator is displayed.

NOTE If you click the Forward wedge once, the model moves forward 50% of the distance between the current location and the pivot point.

- 3 Drag the cursor up or down to change the distance from which you view the model.
- 4 Release the button on your pointing device to return to the wheel.

Look Tool

With the Look tool, you can rotate the current view vertically and horizontally. When rotating the view, your line of sight rotates about the current eye position, like turning your head. The Look tool can be compared to standing in a fixed location, and looking up, down, left or right.

When using the Look tool, you adjust the view of the model by dragging the cursor. As you drag, the cursor changes to the Look cursor and the model rotates around the location of the current view.



In addition to using the Look tool to look around a model, you can also use the tool to transition the current view to a specific face on the model. Press and hold the Shift key before selecting the Look tool on one of the Full Navigation wheels.

Walking Through a Model

When using the **Look** tool from the big **Full Navigation** wheel, you can walk through a model by using the arrow keys on the keyboard. To adjust the walk speed, use the **Options Editor**.

Invert Vertical Axis

When you drag the cursor upward, the target point of the view raises; dragging the cursor downward lowers the target point of the view. To invert the vertical axis for the **Look** tool, use the **Options Editor**.

To look around a view with the Look tool

- 1 Display one of the Full Navigation wheels or the mini Tour Building wheel.
- 2 Click and hold down the Look wedge.
The cursor changes to the Look cursor.
- 3 Drag the pointing device to change the direction in which you are looking.
- 4 Release the button on your pointing device to return to the wheel.

To look at a face in the model with the Look tool

- 1 Display one of the Full Navigation wheels.
- 2 Press and hold down the **SHIFT** key.
- 3 Click and hold down the Look wedge.
The cursor changes to the Look At cursor.
- 4 Drag over the objects in the model until the face highlights that you want to look at.
- 5 Release the button on your pointing device to return to the wheel.

To look around and walk through a model with the Look tool

- 1 Display the big Full Navigation wheel.
- 2 Click and hold down the Look wedge.

The cursor changes to the Look cursor.

- 3 Drag to change the direction in which you are looking.
- 4 While holding down the button on your pointing device, press the arrow keys to walk in the model.
- 5 Release the button on your pointing device to return to the wheel.
- 6 Click Close to exit the wheel.

To invert the vertical axis for the Look tool

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, select the **Invert Vertical Axis** check box in the **Look Tool** area.

Dragging downward and upward lowers and raises the target point of the current view.
- 4 Click **OK**.

Orbit Tool

You use the Orbit tool to change the orientation of a model. The cursor changes to the Orbit cursor. As you drag the cursor, the model rotates around a pivot point while the view remains fixed.



Specify the Pivot Point

The pivot point is the base point used when rotating the model with the Orbit tool. You can specify the pivot point in the following ways:

- **Default pivot point.** When you first open a model, the target point of the current view is used as the pivot point for orbiting the model.
- **Select objects.** You can select objects before the Orbit tool is used to calculate the pivot point. The pivot point is calculated based on the center of the extents of the selected objects.
- **Center tool.** You can specify a point on the model to use as the pivot point for orbiting with the [Center tool](#) (page 247).
- **CTRL+Click and drag.** Press and hold down the CTRL key before clicking the Orbit wedge or while the Orbit tool is active; then drag to the point on the model you want to use as the pivot point. This option is only available when using the big and mini Full Navigation wheels or the mini View Object wheel.

NOTE While the Orbit tool is active, you can press and hold the CTRL key at anytime to move the pivot point used by the Orbit tool. This pivot point is used for subsequent navigation until it is moved.

Maintain Up Direction

You can control how the model orbits around the pivot point by choosing to maintain the up direction of the model. When the up direction is maintained, orbiting is constrained along the XY axis and in the Z direction. If you drag horizontally, the camera moves parallel to the XY plane. If you drag vertically, the camera moves along the Z axis.

If the up direction is not maintained, you can roll the model using the roll ring which is centered around the pivot point. Use the properties dialog box for the SteeringWheels to control whether the up direction is maintained or not for the Orbit tool.



To orbit a model with the Orbit tool

- 1 Display one of the View Object or Full Navigation wheels.
- 2 Click and hold down the Orbit wedge.
The cursor changes to the Orbit cursor.
- 3 Drag to rotate the model.

NOTE Use the Center tool to re-center the model in the current view, if you are using one of the Full Navigation or View Object wheels.

- 4 Release the button on your pointing device to return to the wheel.

To orbit around an object with the Orbit tool

- 1 Press ESC to make sure no commands are active and to clear any previously selected objects.
- 2 Select the objects in the model for which you want to define the pivot point.
- 3 Display one of the View Object or Full Navigation wheels.
- 4 Click and hold down the Orbit wedge.
The cursor changes to the Orbit cursor.
- 5 Drag to rotate the model.
- 6 Release the button on your pointing device to return to the wheel.

To turn on selection sensitivity for the Orbit tool

- 1 Display one of the **View Object** or **Full Navigation** wheels.

- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, select the **Center Pivot on Selection** check box in the **Orbit Tool** section.
- 4 Click **OK**.

The extents of any objects that are selected before the wheel is displayed are used to define the pivot point for the Orbit tool. If no objects are selected, the pivot point used by the Orbit is the one defined by the Center tool.

To maintain the up direction for the Orbit tool

- 1 Display the mini **View Object** wheel or one of the **Full Navigation** wheels.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, select the **Keep Scene Upright** check box in the **Orbit Tool** area.
- 4 Click **OK**.

Orbiting the model is constrained along the XY plane and Z directions.

To roll the model around the pivot point with the Orbit tool

- 1 Display the mini **View Object Wheel** or one of the **Full Navigation** wheels.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, clear the **Keep Scene Upright** check box.
- 4 Click **OK**.
- 5 Click and hold the Orbit wedge.
The cursor changes to the Orbit cursor.
- 6 Press and hold the **SHIFT** key to display the roll ring. Drag to roll the model.
- 7 Release the button on your pointing device to return to the wheel.

To start the Orbit tool with the middle mouse button

- 1 Display one of the wheels other than the big View Object or Tour Building wheels.

- 2 Press and hold down the **SHIFT** key.
- 3 Press and hold down the scroll wheel or middle button on your pointing device and drag to orbit the model.
- 4 Release the button on your pointing device to return to the wheel.

Pan Tool

When the pan tool is active, the Pan cursor (a four-sided arrow) is displayed. Dragging the pointing device moves the model in the same direction. For example, dragging upward moves the model up while dragging downward moves the model down.



TIP If the cursor reaches the edge of the screen, you can continue panning by dragging further to force it to wrap around the screen.

To pan the view with the Pan tool

- 1 Display one of the **Full Navigation** wheels, or the mini **View Object** wheel.
- 2 Click and hold the **Pan** wedge.
The cursor changes to the **Pan** cursor.
- 3 Drag to reposition the model.
- 4 Release the button on your pointing device to return to the wheel.

To start the Pan tool with the middle mouse button

- 1 Display one of the **Full Navigation** wheels, or the mini **View Object** wheel.
- 2 Press and hold down the scroll wheel or middle button.
The cursor changes to the **Pan** cursor.
- 3 Drag to reposition the model.
- 4 Release the wheel or button on your pointing device to return to the wheel.

Rewind Tool

As you use the navigation tools to reorient the view of a model, the previous view is saved to the navigation history. The navigation history holds a representation of the previous views of the model along with a thumbnail. A separate navigation history is maintained for each window; it is not maintained after the window is closed. Rewind navigation history is view-specific.

With the Rewind tool, you can retrieve previous views from the navigation history. From the navigation history, you can restore a previous view or scroll through all of the saved views.

NOTE When you rewind and record a new navigation history, the rewind views are replaced by new views. The navigation history is not saved between sessions.



To restore the previous view

- 1 Display a wheel.
- 2 Click the **Rewind** wedge.

To restore a previous view with the Rewind History panel

- 1 Display a wheel.
- 2 Click and hold the Rewind wedge.
The Rewind History panel is displayed.
- 3 While holding down the button on your pointing device, drag to the left or to the right to restore a previous view.
Dragging to the left restores an older previous view. Dragging to the right restores a view that is newer than the one you are currently viewing. You must have previously used the Rewind tool to see views available on the right. The current position in the navigation history is indicated by the orange box that is dragged along the Rewind History panel.

Up/Down Tool

Unlike the Pan tool, you use the Up/Down tool to adjust the height of the current viewpoint along the model's Z axis. To adjust the vertical elevation of the current view, you drag up or down. As you drag, the current elevation and the allowed range of motion is displayed on a graphical element called the Vertical Distance indicator.

The Vertical Distance indicator has two marks that show the highest (Top) and lowest (Bottom) elevation the view can have. While changing the elevation with the Vertical Distance indicator, the current elevation is shown by the bright orange indicator, while the previous elevation is shown by the dim orange indicator.



To change the elevation of a view

- 1 Display one of the Full Navigation wheels or the Tour Building wheels.

- 2 Click and hold down the Up/Down wedge.
The Vertical Distance indicator is displayed.
- 3 Drag up or down to change the elevation of the view.
- 4 Release the button on your pointing device to return to the wheel.

Walk Tool

With the Walk tool, you can navigate through a model as if you were walking through it. Once you start the Walk tool, the Center Circle icon is displayed near the base of the view and the cursor changes to display a series of arrows. To walk through the model, you drag in the direction in which you want to move in.



Constrain the Walk Angle

When walking through a model, you can constrain the movement angle to the world up vector. If the **Constrain Walk Angle** option is enabled, you can freely walk around while maintaining a constant camera viewpoint elevation; if the walk angle is not constrained, you will “fly” in the direction you are looking. Use the **Options Editor** to constrain the movement angle to the world up vector for the **Walk** tool.

Use Viewpoint Linear Speed

By default, the linear navigation speed in viewpoints is directly related to the size of your model. You can set a specific speed of motion for all viewpoints (**Options Editor** ► **Interface** ► **Viewpoint Defaults**) or for the current

viewpoint (**Viewpoint** tab ► **Save, Load & Playback** panel ► **Edit Current Viewpoint**). Use the **Options Editor** to make the **Walk** tool Use **Viewpoint Linear Speed** settings.

Movement Speed

As you walk or “fly” through a model, you can control the movement speed. Movement speed is controlled by the distance in which the cursor is moved from the **Center Circle** icon and the current movement speed setting. You can adjust the movement speed setting permanently and temporarily as you use the **Walk** tool. To permanently adjust the movement speed, use the **Options Editor** or the < and > keys when the **Walk** tool is active. To temporarily increase movement speed, press and hold the + (plus) key while using the **Walk** tool.

Change the Elevation

As you use the Walk tool, you can adjust the camera elevation by holding down the **SHIFT** key. This temporarily activates the Up/Down tool. With the Up/Down tool active, drag up or down to adjust the elevation of the camera. You can also use the **UP ARROW** and **DOWN ARROW** keys as you walk to adjust the height of the view.

To use the Walk tool to move through the model

- 1 Display one of the Full Navigation wheels or the mini Tour Building wheel.
- 2 Click and hold down the Walk wedge.
The cursor changes to the Walk cursor and the Center Circle icon is displayed.
- 3 Drag in the direction you want to walk.
NOTE While walking, press and hold down the + (plus) key to temporarily increase your movement speed.
- 4 Release the button on your pointing device to return to the wheel.

To change the movement speed for the Walk tool

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, use the **Walk Speed** slider in the **Walk Tool** area.

Dragging the slider to the left decreases the walking speed; dragging the slider to the right increases the walking speed.

- 4 Click **OK**.

To constraint the Walk tool to the world up vector

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, select the **Constrain Walk Angle** check box in the **Walk Tool** area.
- 4 Click **OK**.
Movement when walking is done parallel to the world up of the model.

To make the Walk tool use the viewpoint linear speed

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the **Interface** node, select the **Use Viewpoint Linear Speed** check box in the **Walk Tool** area.
- 4 Click **OK**.

To adjust the height of the current view from the Walk tool

- 1 Display one of the Full Navigation wheels or the mini Tour Building wheel.
- 2 Click and hold down the Walk wedge.
The cursor changes to the Walk cursor and the Center Circle icon is displayed.
- 3 Do one of the following:
 - Press and hold down the **SHIFT** key to enable the Up/Down tool; drag up or down.
 - Press and hold down the **UP ARROW** or **DOWN ARROW** key.
- 4 Release the button on your pointing device to return to the wheel.

Zoom Tool

You use the **Zoom** tool to change the zoom magnification of a model.



NOTE When you start the **Zoom** tool from the **Full Navigation** wheel, incremental zooming must be enabled in the **Options Editor** in order to use CTRL+click and SHIFT+click.

Zoom Constraints

When changing the magnification of a model with the Zoom tool, you cannot zoom in any further than the focus point or out past the extents of the model. The direction you can zoom in and out is controlled by the center point set by the Center tool.

NOTE Unlike the Zoom tool on the big View Object wheel, the Zoom tool on the mini View Object wheel and the Full Navigation wheels are not constrained.

To zoom the view with a single click

Note: You must enable incremental zoom when using the **Full Navigation** wheels, or a mini **View Object** wheel. For the big **View Object** wheel, the incremental zoom is always enabled.

- 1 Do the following to make sure the **Enable Single-Click Incremental Zoom-In** option is selected:
 - 1 Display the **Full Navigation** wheel.
 - 2 Right-click the wheel, and click **SteeringWheel Options**.
 - 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select the **Enable Single-Click Incremental Zoom-In** check box in the **Zoom Tool** area.

- 4 Click **OK**.
- 2 Display a wheel that has the **Zoom** tool.
- 3 Click the Zoom wedge.

The magnification of the model is increased and you are zoomed in closer to the model. If you hold down the **SHIFT** key while clicking the Zoom wedge, the model is zoomed out; you can hold down the **CTRL** key to zoom in.

To zoom a view in and out by dragging

- 1 Display one of the **Full Navigation** wheels, or one of the **View Object** wheels.
- 2 Click and hold down the Zoom wedge.

The cursor changes to the Zoom cursor.
- 3 Drag vertically to zoom in or out.
- 4 Release the button on your pointing device to return to the wheel.

To zoom into an area of the model by specifying window

- 1 Display one of the Full Navigation wheels or the mini View Object wheel.
- 2 Press and hold down the **SHIFT** key.
- 3 Click and hold down the Zoom wedge.

The cursor changes to the Zoom cursor.
- 4 Drag the pointing device to define the opposite corner of the window that defines the area in which you want to zoom.

NOTE Holding down the **CTRL** key while defining the second point of the window determines if the first point of the window is used as the corner or center of the window being dragged. When the **CTRL** key is held down, the first point defines the center of the window.

- 5 Release the button on your pointing device to return to the wheel.

To zoom in and out by scrolling the mouse wheel when the SteeringWheels is displayed

- 1 Display one of the wheels other than the big Tour Building wheel.
- 2 Scroll the wheel forward or backward to zoom in or out.
- 3 Release the button on your pointing device to return to the wheel.

You use the Zoom tool to change the zoom magnification of a model. The following mouse click and key combinations are available to control how the Zoom tool behaves:

- **Click.** If you click the **Zoom** tool on a wheel, the current view is zoomed in by a factor of 25 percent. If you are using the **Full Navigation** wheel, incremental zoom must be enabled in the **Options Editor**.
- **SHIFT+click.** If you hold down the **SHIFT** key before you click the **Zoom** tool on a wheel, the current view is zoomed out by a factor of 25 percent. Zooming is performed from the location of the cursor, and not the current pivot point.
- **CTRL+click.** If you hold down the **CTRL** key before you click the **Zoom** tool on a wheel, the current view is zoomed in by a factor of 25 percent. Zooming is performed from the location of the cursor, and not the current pivot point.
- **Click and drag.** If you click the **Zoom** tool and hold down the button on your pointing device, you can adjust the magnification of the model by dragging up and down.
- **CTRL+click and drag.** When using the Full Navigation wheels or the mini View Object wheel, you can control the target point used by the **Zoom** tool. By holding down the **CTRL** key, the **Zoom** tool uses the location of the previous pivot point defined by the **Zoom**, **Orbit**, or **Center** tool.
- **SHIFT+click and drag.** When using the Full Navigation wheels or the mini View Object wheel, you can zoom in to an area of the model by dragging a rectangular window around the area you want to fit in the window. Hold down the **SHIFT** key and then click and drag a window around the area in which you want to zoom.

NOTE If you hold down the **CTRL** key along with the **SHIFT** key, you can zoom in to an area of a model using a center-based window instead of one defined by opposite corners.

- **Mouse wheel.** When a wheel is displayed, scroll the mouse wheel up or down to zoom the view of the model in or out.

NOTE When you use the **Zoom** tool from the Full Navigation wheel or the View Object wheel, the point in the view where you click to zoom becomes the **Center** point for future **Orbit** operations until you either use the **Zoom** tool again or use the **Center** tool. If you press **CTRL** before you click the **Zoom** wedge, the **Center** point does not change.

Classic Navigation Modes and Tools

In the **Classic** user interface, there are nine navigation modes available from the **Navigation Mode** toolbar to control how you move around the **Scene View**: six camera-centric modes and three model-centric modes.



TIP You can use some of these classic navigation modes with the standard (ribbon) user interface. The **Options Editor** allows you to select between old and new modes.

In a camera-centric mode, the camera moves within the scene, whereas in a model-centric mode, model moves inside the scene. For example, orbit and examine modes essentially do the same thing, except that orbit mode moves the camera around the focal point and examine mode moves the model around the focal point.

NOTE Navigation modes and [SteeringWheels](#) (page 287) are mutually exclusive, so activating navigation mode deactivates the current SteeringWheel menu.

Movement in each mode is based on the keyboard arrow keys, the SHIFT and CTRL keys, and mouse drags. The mouse wheel is also supported, allowing quick and easy zooming or tilting, depending on the current navigation mode.

TIP Dragging with the left mouse button while holding down the CTRL key performs the same actions as dragging with the middle mouse button, which is useful if you only have a two-button mouse.

The SHIFT and CONTROL keys modify the movement, for example holding down SHIFT in walk mode speeds up movement, and holding down CTRL in this mode, glides the camera left/right and up/down.

NOTE Gliding the camera is opposite to panning the model. Gliding is a camera-centric motion and panning is a model-centric motion.

Mode	Description
	Walk. Enables you to walk through the model on a horizontal plane ensuring that “up” is always “up”.

Mode	Description
	Look Around. Enables you to look around the model from the current camera position and gives the effect that you are moving your head around.
	Zoom. Enables you to zoom into and out of the model. Cursor up zooms in and cursor down zooms out.
	Zoom to a Box. Enables you to drag a box so that the contents of the box fill the view.
	Pan. Enables you to pan the model rather than the camera.
	Orbit. Enables you to orbit the camera around the model, ensuring that “up” is always “up”. The camera always orbits around the focal point of the model.
	Examine. Enables you to rotate the model about.
	Fly. Enables you to fly around the model like in a flight simulator.
	Turntable. Enables you to spin the model around the up vector. This navigation mode behaves as though the model is sitting on a turntable, ensuring that “up” is always “up”.

Walk Mode

In **Walk** mode, you can navigate through a model as if you were walking through it. In this mode, the up direction is always maintained.

Once you start walk mode, the cursor changes to the walk cursor. To walk through the model, you drag in the direction in which you want to move in.



To use walk mode to move through the model

- 1 Click **Walk**  on the **Navigation Mode** toolbar.
- 2 To move, hold down the left mouse button as you drag in the direction you want to walk, or use the cursor keys. The camera spins left and right, and moves forwards and backwards.

NOTE Holding down the SHIFT key speeds up this movement.

- 3 To glide, hold down the CTRL key as you drag the mouse. The camera glides left and right and up and down.
As walk mode is camera-centric, this mode differs from the normal pan mode in that the camera is moved rather than the model.
- 4 To tilt the camera up and down, spin the mouse wheel.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Mode** ► **Walk**
 **Command entry:** CTRL + 2

Look Around Mode

In **Look Around** mode, you can rotate the current view vertically and horizontally. When rotating the view, your line of sight rotates about the current eye position, like turning your head.



To look around a view

- 1 Click **Look Around**  on the **Navigation Mode** toolbar.
- 2 To look around, drag the left mouse button, or use the cursor keys. The camera looks left, right, up or down.

NOTE Holding down the SHIFT key speeds up this movement.

- 3 To rotate the camera around its viewing axis, hold down the CTRL key.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Mode** ► **Look Around**

 **Command entry:** CTRL + 3

Zoom Mode

In **Zoom** mode, you can zoom into and out of the model.



To zoom the view

- 1 Click **Zoom**  on the **Navigation Mode** toolbar.
- 2 Drag the left mouse button up and down, or use the up and down cursor keys, to zoom in and out respectively.

 **Menu:** Classic user interface: **Viewpoint** > **Navigation Mode** > **Zoom**
 **Command entry:** CTRL + 4

Zoom Box Mode

In **Zoom Box** mode, you can zoom into an area of the model by dragging a rectangular box around the area you want to fit in the **Scene View**.



To zoom in to an area of the model by specifying box

- 1 Click **Zoom Box**  on the **Navigation Mode** toolbar.
- 2 Drag a box with the left mouse button over the **Scene View** to fill the view with the contents of the box.

NOTE Holding down the SHIFT or CTRL keys, or spinning the mouse wheel, temporarily enables normal **Zoom** mode.

 **Menu:** Classic user interface: **Viewpoint** > **Navigation Mode** > **Zoom Window**
 **Command entry:** CTRL + 5

Pan Mode

In **Pan** mode, you can move the model, rather than the camera. For example, dragging upward moves the model up while dragging downward moves the model down.



To pan a model

- 1 Click **Pan**  on the **Navigation Mode** toolbar.
- 2 Drag the left mouse button to pan the model up, down, left and right.

NOTE Holding down the SHIFT or CTRL keys, or spinning the mouse wheel, temporarily enables normal **Zoom** mode.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Mode** ► **Pan**
 **Command entry:** CTRL + 6

Orbit Mode

In **Orbit** mode, the camera moves around the focal point of the model. In this mode, the up direction is always maintained.



To orbit a model

- 1 Click **Orbit**  on the **Navigation Mode** toolbar.
- 2 To rotate the camera around the model, drag the left mouse button, or using the cursor keys.

NOTE Holding down the SHIFT key, or spinning the mouse wheel, temporarily enables normal **Zoom** mode.

- 3 To glide the camera, hold down the CTRL key as you drag the mouse. The camera glides left and right and up and down.
As orbit mode is camera-centric, this mode differs from normal pan mode in that the camera is moved rather than the model.

 **Menu:** Classic user interface: **Viewpoint > Navigation Mode > Orbit**

 **Command entry:** CTRL + 7

Examine Mode

In **Free Orbit** mode, you can rotate the model around the focal point.



To examine a model

- 1 Click **Examine**  on the **Navigation Mode** toolbar.
- 2 To rotate the model around the focal point, drag the left mouse button, or using the cursor keys. If the mouse is moving when you let go of the button, the model keeps spinning. Clicking on the model stops this.

NOTE Holding down the SHIFT key, or spinning the mouse wheel, temporarily enables normal **Zoom** mode. Holding down the CTRL key, temporarily enables normal **Pan** mode.

 **Menu:** Classic user interface: **Viewpoint > Navigation Mode > Examine**

 **Command entry:** CTRL + 8

Fly Mode

In **Fly** mode, you can move around the model like in a flight simulator.



To use Fly mode to move through the model

- 1 Click **Fly**  on the **Navigation Mode** toolbar.
- 2 Hold down the left mouse button to move the camera forward. As in a flight simulator, the left mouse button banks left/right when dragged left or right and tilts up/down when dragged up or down.

NOTE Holding down the SHIFT key speeds up this movement.

- 3 Use the up and down cursor keys to zoom the camera in and out respectively; use the left and right cursor keys to spin the camera left and right respectively.

NOTE Holding down the CTRL key rotates the camera around its viewing axis, while still moving forward.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Mode** ► **Fly**
 **Command entry:** CTRL + 9

Turntable Mode

In **Turntable** mode, you can spin the model around the up vector as though the model is sitting on a turntable. In this mode, the up direction is always maintained.



To spin model on a turntable

- 1 On the navigation bar, click **Turntable**  on the **Navigation Mode** toolbar.
- 2 Drag the left mouse button left and right, or use the left and right cursor keys, to spin the turntable left and right respectively.

NOTE Holding down the SHIFT key or spinning the mouse wheel, temporarily enables normal **Zoom** mode. Holding down the CTRL key, temporarily enables normal **Pan** mode.

- 3 To tilt the turntable up and down, spin the mouse wheel, or use the up and down cursor keys.

 **Menu:** Classic user interface: **Viewpoint > Navigation Mode > Turntable**

 **Command entry:** CTRL + 0

View All Tool

Makes the complete model fit into the Scene View.

Using this function dollies and pans the camera so that the entire model is shown in the current view, which is very useful if you get lost inside a model or lose it completely.

Occasionally, you may get a blank view. This is usually because there are items that are very small in comparison to the main model, or items that are located a long way away from the main model. In these cases, right-click an item in the **Selection Tree** and click **Zoom Selected** to find your way back to the model before trying to figure out which items are “lost”.

To view everything

- 1 Click **View All**  on the **Navigation Tools** toolbar.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Tools** ► **View All**

Shortcut menu: **Scene** ► **View All**

Zoom Selected Tool

Zooms the camera so that the selected items fill the Scene View.

To view selected items

- 1 Click **Zoom Selected**  on the **Navigation Tools** toolbar.

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Tools** ► **Zoom Selected**

Shortcut menu: **Scene** ► **Zoom Selected**

ViewCube

Autodesk® ViewCube® navigation tool provides visual feedback of the current orientation of a model. You can use the ViewCube tool to adjust the viewpoint of your model. The ViewCube is not available in a 2D workspace.

Overview of ViewCube

The ViewCube tool is a persistent, clickable, and draggable interface that you use to switch between views of your model.

When you display the ViewCube tool, by default it is shown in the top-right corner of the **Scene View** over the model in an inactive state. The ViewCube tool provides visual feedback about the current viewpoint of the model as view changes occur. When the cursor is positioned over the ViewCube tool, it becomes active. You can drag or click the ViewCube, switch to one of the available preset views, roll the current view, or change to the Home view of the model.



TIP When the navigation bar is linked to the ViewCube, both can be moved around the **Scene View**. See [Reposition and Reorient the Navigation Bar](#) (page 285) for more information.

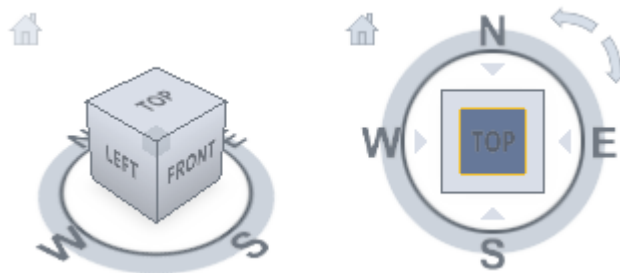
Control the Appearance of ViewCube

The ViewCube tool is displayed in one of two states: inactive and active. When the ViewCube tool is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the model.

In addition to controlling the opacity level of the ViewCube when it is inactive, you can also control its size, and the display of the compass. The settings used to control the appearance of the ViewCube are located in the **Options Editor**.

Use the Compass

The compass is displayed below the ViewCube tool and indicates which direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model, or you can click and drag one of the cardinal direction letters or the compass ring to interactively rotate the model around the pivot point.



Drag or Click ViewCube

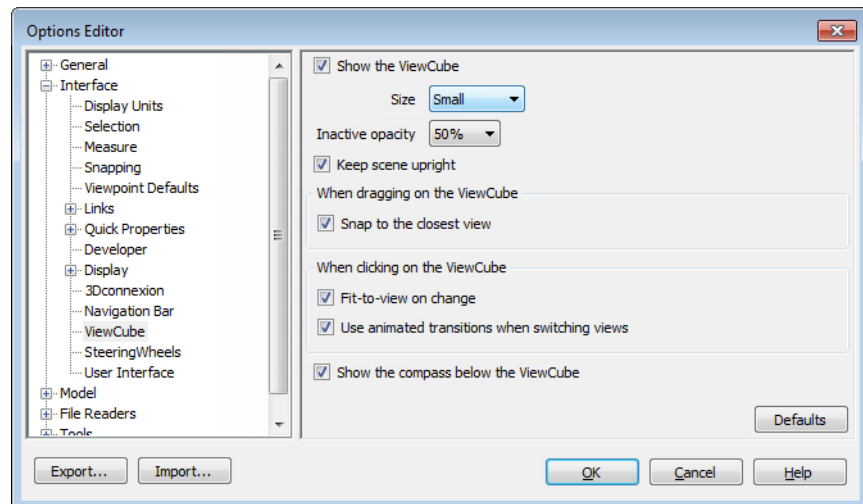
When you drag or click the ViewCube tool, the view of the model reorients around a pivot point. The pivot point is displayed at the center of the object that was last selected before using the ViewCube tool.

To display or hide the ViewCube

- Click **View** tab ► **Navigation Aids** panel ► **ViewCube**  .

To control the size of the ViewCube

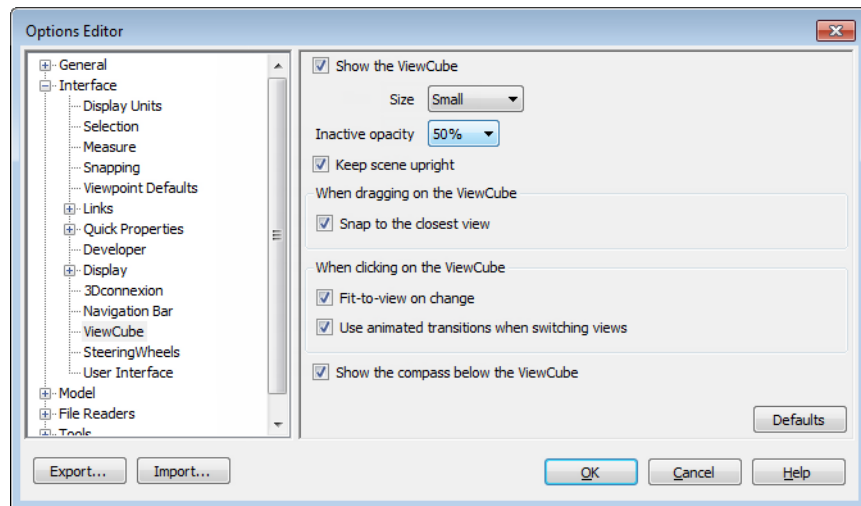
- 1 Right-click the ViewCube tool, and click **ViewCube Options**.
- 2 In the **Options Editor**, the **ViewCube** page under the **Interface** node, select an option from the **Size** drop-down list.



- 3 Click **OK**.

To control the inactive opacity of the ViewCube

- 1 Right-click the ViewCube tool, and click **ViewCube Options**.
- 2 In the **Options Editor**, the **ViewCube** page under the **Interface** node, select an option from the **Inactive Opacity** drop-down list.



- 3 Click **OK**.

To display the compass for the ViewCube

- 1 Right-click the ViewCube tool, and click **ViewCube Options**.
- 2 In the **Options Editor**, the **ViewCube** page under the **Interface** node, select **Show Compass Below the ViewCube**.
- 3 Click **OK**.

The compass is displayed below the ViewCube tool and indicates the direction of north for the model.

ViewCube Menu

Use the ViewCube menu to restore and define the Home view of a model, switch between view projection modes, and change the interactive behavior and appearance of the ViewCube tool.

To display the ViewCube menu

To display the ViewCube menu, do one of the following:

- Right-click on the compass, Home icon, or the main area of the ViewCube tool.
- Click the context menu button located below the ViewCube tool.

The ViewCube menu has the following options:

- **Home.** Restores the Home view saved with the model. This view is in synchronization with the Go Home view option in the SteeringWheels menu.
- **Perspective.** Switches the current view to perspective projection.
- **Orthographic.** Switches the current view to orthographic projection.
- **Lock to Selection.** Uses the selected objects to define the center of the view when a view orientation change occurs with the ViewCube tool.

NOTE If you click Home on the ViewCube tool, the view returns to the Home view even if Lock to Current Selection is selected.

- **Set Current View as Home.** Defines the Home view of the model based on the current view.
- **Set Current View as Front.** Defines the Front view of the model.
- **Reset Front.** Resets the Front view of the model to its default orientation.
- **ViewCube Options.** Displays the **Options Editor** where you can adjust the appearance and behavior of the ViewCube tool.
- **Help.** Launches the online Help system and displays the topic for the ViewCube tool.

Reorient the View of a Model with ViewCube

ViewCube is used to reorient the current view of a model. You can reorient the view of a model with the ViewCube tool by clicking pre-defined areas to set a preset view current, click and drag to freely change the view angle of the model, and define and restore the Home view.

Reorient the Current View

The ViewCube tool provides twenty-six defined parts to click and change the current view of a model. The twenty-six defined parts are categorized into three groups: corner, edge, and face. Of the twenty-six defined parts, six of the parts represent standard orthogonal views of a model: top, bottom, front, back, left, and right. Orthogonal views are set by clicking one of the faces on the ViewCube tool.

NOTE When the cursor is over one of the clickable areas of the ViewCube tool, the cursor changes to an arrow with a small cube to indicate that it is over the ViewCube tool. A tooltip is also displayed. The tooltip describes the action that you can perform based on the location of the cursor over the ViewCube tool.

You use the other twenty defined parts to access angled views of a model. Clicking one of the corners on the ViewCube tool reorients the current view of the model to a three-quarter view, based on a viewpoint defined by three sides of the model. Clicking one of the edges reorients the view of the model to a half view based on two sides of the model.

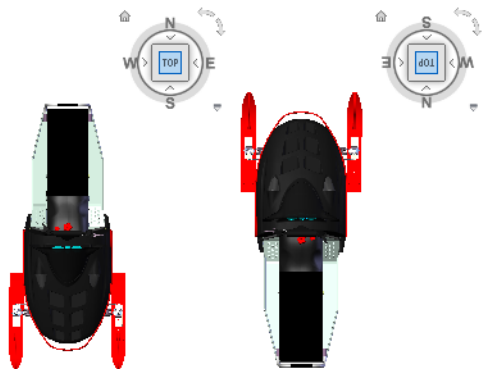


You can also click and drag the ViewCube tool to reorient the view of a model to a custom view other than one of the twenty-six predefined parts. As you drag, the cursor changes to indicate that you are reorienting the current view of the model. If you drag the ViewCube tool close to one of the preset orientations and it is set to snap to the closest view, the ViewCube tool rotates to the closest preset orientation.

The outline of the ViewCube tool helps you identify the form of orientation it is in: standard or fixed. When the ViewCube tool is in standard orientation, not orientated to one of the twenty-six predefined parts, its outline is displayed as dashed. The ViewCube tool is outlined in a solid continuous line when it is constrained to one of the predefined views.

Roll a Face View

When you view a model from one of the face views, two roll arrow buttons are displayed near the ViewCube tool. Use the roll arrows to rotate the current view 90 degrees clockwise or counterclockwise around the center of the view.



Switch to an Adjacent Face

When the ViewCube tool is active while viewing a model from one of the face views, four orthogonal triangles are displayed near the ViewCube tool. You use these triangles to switch to one of the adjacent face views.



Front View

You can define the Front view of a model to define the direction of the face views on the ViewCube tool. Along with the Front view, the up direction of a model is also used to define the direction of the face views on the ViewCube tool.

NOTE Front view is a global setting and will be the same for viewpoints.

To reorient the current view to a preset orientation

- Click one of the faces, edges, or corners on the ViewCube tool.

To view an adjacent face

NOTE: Make sure a face view is current.

- Click one of the triangles displayed near the edges of the ViewCube tool.

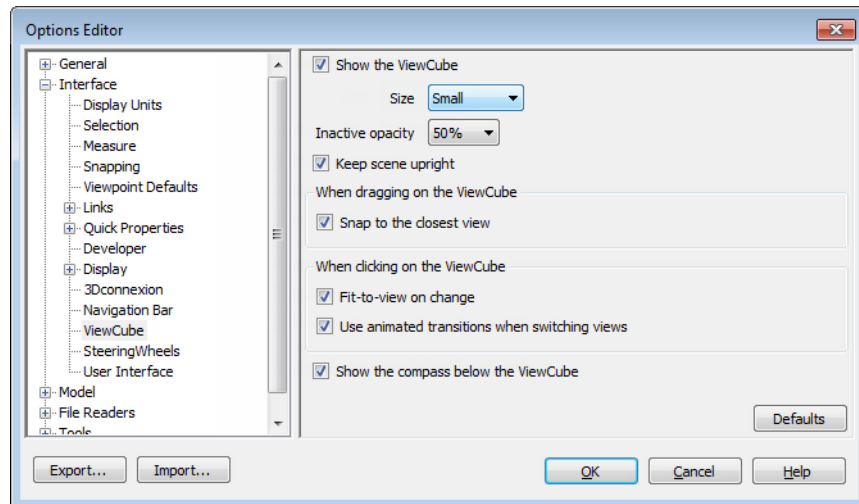


To interactively reorient the view

- Click the ViewCube tool, hold down the left mouse button, and drag in the direction that you want to orbit the model.

To use animated transitions when reorienting a view to a preset orientation

- 1 Right-click the ViewCube tool, and click **ViewCube Options**.
- 2 In the **Options Editor**, the **ViewCube** page under the **Interface** node, select **Use Animated Transitions When Switching Views**.



When checked, transitions from one view to another appear animated when clicking a predefined area on the ViewCube tool.

- 3 Click **OK**.

To automatically fit the model after a view orientation

- 1 Right-click the ViewCube tool, and click **ViewCube Options**.
- 2 In the **Options Editor**, the **ViewCube** page under the **Interface** node, select **Fit-to-View on Change**.
- 3 Click **OK**.

To roll a face view

NOTE: Make sure a face view is displayed.

- Click one of the roll arrows displayed above and to the right of the ViewCube tool.

The left roll arrow rotates the view 90 degrees counterclockwise; the right roll arrow rotates the view 90 degrees clockwise.

To define the front view

- Right-click the ViewCube tool, and click Set Current View as Front.

To restore the Front view

- Right-click the ViewCube tool, and click Reset Front.

Set the View Projection Mode

The ViewCube tool supports two view projection modes (**Perspective** and **Orthographic**). **Orthographic** projection is also referred to as parallel projection. **Perspective** projected views are calculated based on the distance from a theoretical camera and target point. The shorter the distance between the camera and the target point, the more distorted the perspective effect appears; greater distances produce less distorted affects on the model. **Orthographic** projected views display all the points of a model being projected parallel to the screen.

Orthographic projection mode makes it easier to work with a model due to all the edges of the model appearing as the same size, regardless of the distance from the camera. Orthographic projection mode though, is not how you commonly see objects in the real world. Objects in the real world are seen in

perspective projection. So when you want to generate a rendering or hidden line view of a model, using perspective projection will give the model a more realistic look.

The following illustration shows the same model viewed from the same viewing direction, but with different view projections.



To change the view projection mode

- Right-click the **ViewCube** tool, and click one of the following options:
 - **Orthographic**
 - **Perspective**

Home View

The Home view is a special view stored with a model that makes it easy to return to a known or familiar view. You can define any view of the model as the Home view. The saved Home view can be applied to the current view by clicking the Home button above the ViewCube tool or from the ViewCube menu.

To define the Home view

- Right-click the ViewCube tool, and click **Set Current View as Home**.

 **Command entry:** CTRL+SHIFT+Home

To reorient the model to the Home view

- Click the Home button () located near the ViewCube tool.
- Right-click the ViewCube tool, and click Home.

Examine Individual Objects with ViewCube

You can lock the ViewCube tool to a set of selected objects. Locking a selection of objects to the ViewCube tool defines the center of the current view and the distance from center for the view based on the selected objects. To turn off **Lock to Selection**, you can click the **Lock to Selection**  button next to the Home view button.

Selecting and deselecting objects after Lock to Selection is turned on has no effect on the center or distance from the center of the view when a view orientation changes. You cannot zoom fit to view a model when Lock to Selection is on, even if the ViewCube tool is set to zoom fit to view after each view orientation change.

To lock to the current selection

- Right-click the ViewCube tool, and click Lock to Selection.

If Lock to Selection is checked when a view orientation change occurs, the selected objects are used to calculate the center of the view and the view zooms to the extents of the selected objects. When cleared, the selected objects are used to calculate the center of the view and the view zooms to the extents of the model.

To examine an individual object with ViewCube

- 1 In the model, select one or more objects to define the centerpoint of the view.
- 2 Click one of the preset locations on the ViewCube tool, or click and drag the ViewCube tool to reorient the view of the model.

The ViewCube tool reorients the view of the model based on the centerpoint of the selected objects.

Navigation Bar

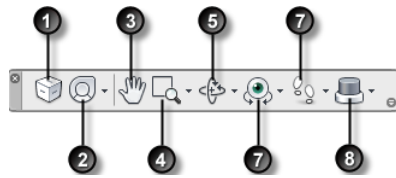
Unified and product-specific navigation tools can be accessed from the navigation bar.

Overview of Navigation Bar

The navigation bar is a user interface element where you can access both unified and product-specific navigation tools.

Unified navigation tools (such as Autodesk® ViewCube®, 3Dconnexion®, and SteeringWheels®) are those that can be found across many Autodesk products. Product-specific navigation tools are unique to a product. The navigation bar floats over and along one of the sides of the **Scene View**.

You start navigation tools by clicking one of the buttons on the navigation bar or selecting one of the tools from a list that is displayed when you click the smaller portion of a split button.



1. [ViewCube](#) (page 273) Indicates the current orientation of a model, and is used to reorient the current view of a model. Clicking this button displays the ViewCube in the Scene View when it's not visible.

2. [SteeringWheels](#) (page 287) Collection of wheels that offer rapid switching between specialized navigation tools.

3. [Pan tool](#) (page 243). Activates the pan tool and moves the view parallel to the screen.

4. [Zoom tools](#) (page 243). Set of navigation tools for increasing or decreasing the magnification of the current view of the model.

5. [Orbit tools](#) (page 244). Set of navigation tools for rotating the model around a pivot point while the view remains fixed.

6. [Look tools](#) (page 245). Set of navigation tools for rotating the current view vertically and horizontally.

7. [Walk and Fly tools](#) (page 246). Set of navigation tools for moving around the model and controlling realism settings.

8. [3Dconnexion](#) (page 298) Set of navigation tools used to reorient the current view of a model with a 3Dconnexion 3D mouse.

NOTE In a 2D workspace, only the 2D navigation tools (such as 2D SteeringWheels, Pan, Zoom, and the 2D Mode 3Dconnexion tools) are accessible.

To display or hide the navigation bar

■ Click **View** tab ► **Navigation Aids** panel ► **Navigation Bar**  .

Reposition and Reorient the Navigation Bar

The position and orientation of the navigation bar can be adjusted by linking it to the ViewCube tool, docking it when the ViewCube tool is not displayed, or freely positioning it along one of the edges of the current window.

When linked to the ViewCube tool, the navigation bar is positioned below the ViewCube tool and in a vertical orientation. When not linked or docked, the navigation bar can be freely aligned along one of the edges of the **Scene View**.

You can specify how the navigation bar can be repositioned from the **Customize** menu. When the navigation bar is not linked to the ViewCube tool or docked, a grip handle is displayed. Drag the grip handle on the navigation bar to reposition it along one of the sides of the **Scene View**.



If the side of the **Scene View** that the navigation bar is aligned to is not long enough to show the entire navigation bar, it is truncated to fit. When truncated, the **More Controls**  button is displayed and replaces the **Customize**  button. When you click the **More Controls** button, a menu is displayed that contains the navigation tools that are not currently being displayed.

To reposition the navigation bar and ViewCube

- 1 On the navigation bar, click **Customize**.
- 2 Click **Customize** menu ► **Docking Positions** ► check **Link to ViewCube**.

When **Link to ViewCube** is checked, both the navigation bar and ViewCube are repositioned together around the current window. When ViewCube is not displayed, the navigation bar is docked in the same location in which ViewCube would be instead.

- 3 Click **Customize** menu ► **Docking Positions** ► and then a docking position.

The navigation bar and ViewCube are repositioned.

To link the position of the navigation bar to ViewCube

- 1 On the navigation bar, click Customize.
- 2 Click Customize menu ➤ Docking Positions ➤ check Link to ViewCube.

When Link to ViewCube is checked, both the navigation bar and ViewCube are repositioned together around the current window.

To freely reposition the navigation bar along the edge of the current window

TIP In a 2D workspace, click the grip handle and drag the navigation bar to reposition it.

- 1 On the navigation bar, click Customize.
- 2 Click Customize menu ➤ Docking Positions ➤ uncheck Link to ViewCube.

The grip handle for the navigation bar is displayed along the top of the navigation bar.

- 3 Click the grip handle and drag the navigation bar along the edge of the window where you want it displayed. Release the button on the pointing device to orient the navigation bar along the edge of the window.
- 4 Drag the navigation bar along the window's edge to adjust its position along the window's edge.

Control the Display of Navigation Tools on the Navigation Bar

You can control which unified and product-specific navigation tools are displayed on the navigation bar with the **Customize** menu.

The Customize menu is displayed by clicking the Customize button on the lower-right side of the navigation bar. From the Customize menus, you click the navigation tools that you want displayed on the navigation bar. The position of the navigation tools on the navigation bar is predefined and cannot be changed.

NOTE In a 3D workspace, the ViewCube button is displayed on the navigation bar only when the ViewCube tool is hidden in the **Scene View**. In a 2D workspace, the ViewCube button is not available.

To customize the navigation bar

- 1 On the navigation bar, click Customize.
- 2 On the Customize menu, click the navigation tool you want to display on the navigation bar.
A check mark next to a navigation tool's name indicates it is displayed on the navigation bar. Uncheck the navigation tool to remove it from the navigation bar.

You can display a shortcut menu for tools on the navigation bar by right-clicking them. The following commands are available on the shortcut menu whenever they are applicable:

Command	Purpose
Remove from Navigation Bar	Removes the tool from the navigation bar. This is equivalent to unchecking the relevant check box in the Customize menu.
Close Navigation Bar	Hides the navigation bar.

SteeringWheels

SteeringWheels™ are tracking menus that follow your cursor, and from which you can access different 2D and 3D navigation tools from a single tool.

Overview of SteeringWheels

SteeringWheels, also known as wheels, can save you time by combining many of the common navigation tools into a single interface. Wheels are specific to the context from which a model is being viewed.

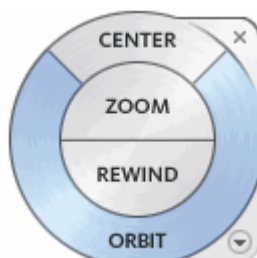
The following illustrations show the different wheels available:



2D Navigation Wheel



Full Navigation Wheel



View Object Wheel (Basic Wheel)



Tour Building Wheel (Basic Wheel)



Pan

Mini Full Navigation Wheel



Pan

Mini View Object Wheel



Up/Down

Mini Tour Building Wheel

NOTE SteeringWheels and [classic navigation modes](#) (page 264) are mutually exclusive, so activating a SteeringWheel deactivates the currently selected **Classic** navigation mode.

Display and Use Wheels

Pressing and dragging on a wedge of a wheel is the primary mode of interaction. After a wheel is displayed, click one of the wedges and hold down the button on the pointing device to activate the navigation tool. Drag to reorient the current view. Releasing the button returns you to the wheel.

Control the Appearance of Wheels

You can control the appearance of the wheels by switching between the different styles of wheels that are available, or by adjusting the size and opacity. Wheels are available in two different styles: big and mini. The big wheel is larger than the cursor, and labels are shown on the wheel wedges. The mini wheel is about the same size as the cursor, and labels are not displayed on the wheel wedges.



Big Full Navigation Wheel



Mini Full Navigation Wheel

The size of a wheel controls how large or small the wedges and labels appear on the wheel; the opacity level controls the visibility of the objects in the model behind the wheel.

Control Tooltips for Wheels and Messages for Tools

Tooltips are displayed for each button on a wheel as the cursor is moved over them. The tooltips appear below the wheel and identify what action will be performed if the wedge or button is clicked.

Similar to tooltips, tool messages and cursor text are displayed when you use one of the navigation tools from a wheel. Tool messages are displayed when a navigation tool is active; they provide basic instructions about using the tool. Tool cursor text displays the name of the active navigation tool near the cursor. Disabling tool messages and cursor text only affects the messages that are displayed when using the mini wheels or the big Full Navigation wheel.

To display a wheel

- 1 On the navigation bar, click the arrow below the SteeringWheels button.
- 2 Click the wheel you want to display, for example **Full Navigation Wheel**.

Ribbon: Viewpoint tab ► Navigate panel ► SteeringWheels 

 **Toolbar: Classic** user interface: Navigation Mode ► SteeringWheels 

To close a wheel

- Press **SHIFT+W**

To change the size of a wheel

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select an option from the **Size** drop-down list in the **Big Wheels** or **Mini Wheels** area.
- 4 Click **OK**.

To change the opacity of a wheel

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select an option from the **Opacity** drop-down list in the **Big Wheels** or **Mini Wheels** area.
- 4 Click **OK**.

To enable tooltips for wheels

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select the **Show Tooltips** check box in the **On-Screen Messages** area.
Tooltips are displayed for each wedge and button on a wheel when the cursor moves over the wheel.
- 4 Click **OK**.

To enable tool messages for wheels

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.

- 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select the **Show Tool Messages** check box in the **On-Screen Messages** area.

Messages are displayed when you use the navigation tools.

- 4 Click **OK**.

To enable tool cursor text for wheels

- 1 Display a wheel.
- 2 Right-click the wheel, and click **SteeringWheels Options**.
- 3 In the **Options Editor**, the **SteeringWheels** page under the Interface node, select the **Show Tool Cursor Text** check box in the **On-Screen Messages** area.

The name of the active tool is displayed near the cursor when the tool is in use.

- 4 Click **OK**.

Wheel Menu

From the Wheel menu, you can switch between different wheels and change the behavior of some of the navigation tools on the current wheel.

Use the Wheel menu to switch between the big and mini wheels that are available, go to the Home view, change the preferences of the current wheel, and control the behavior of the orbit, look, and walk 3D navigation tools. The menu items available on the Wheel menu are dependent on the current wheel and program.

To display the Wheel menu

- Click the down arrow in the lower-right corner of the wheel or right-click on the wheel.

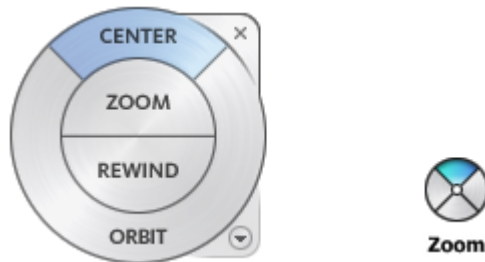
The Wheel menu has the following options:

- **Basic View Object Wheel.** Displays the big View Object wheel.
- **Basic Tour Building Wheel.** Displays the big Tour Building wheel.
- **Full Navigation Wheel.** Displays the big Full Navigation wheel.

- **Advanced Wheels.** Displays the mini View Object, Tour Building, or Full Navigation wheel.
- **Home.** Goes to the Home view saved with the model.
 NOTE This is the Home view as set using the ViewCube.
- **Fit to Window.** Resizes and centers the current view to display all objects in the **Scene View**. This is equivalent to clicking **View All**  on the **Navigation Tools** toolbar in the **Classic** user interface.
- **Restore Original Center.** Restores the center point of the view to the extents of the model.
- **Level Camera.** Rotates the current view so it is relative to the XY ground plane.
- **SteeringWheels Options.** Displays the **Options Editor** where you can adjust the appearance and behavior of SteeringWheels.
- **Help.** Launches the online Help system and displays the topic for SteeringWheels.
- **Close Wheel.** Closes the wheel.

View Object Wheels

With the View Object wheels (big and mini), you can view individual objects or features in a model. The big View Object wheel is optimized for new 3D users while the mini View Object wheel is optimized for experienced 3D users.



Big View Object Wheel

The big View Object wheel wedges have the following options:

- **Center (page 247)**. Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- **Zoom (page 261)**. Adjusts the magnification of the current view.
- **Rewind (page 256)**. Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.
- **Orbit (page 251)**. Rotates the current view around a fixed pivot point at the view's center.

Mini View Object Wheel

The mini View Object wheel wedges have the following options:

- **Zoom (Top wedge) (page 261)**. Adjusts the magnification of the current view.
- **Rewind (Right wedge) (page 256)**. Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Pan (Bottom wedge) (page 243)**. Repositions the current view by panning.
- **Orbit (Left wedge) (page 251)**. Rotates the current view around a fixed pivot point.

NOTE When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

To switch to the mini View Object wheel

- Right-click the wheel, and click **Advanced Wheels > Mini View Object Wheel**.

 **Toolbar:** Navigation bar > SteeringWheels > Mini View Object Wheel

Ribbon: Viewpoint tab > Navigate panel > Steering WheelsMini

View Object 

 **Menu:** Classic user interface: View > SteeringWheels > Mini View Object Wheel

 **Toolbar: Classic** user interface: Navigation Mode ► Mini View Object Wheel 

To switch to the big View Object wheel

- Right-click the wheel, and click **Basic View Object Wheel**.

 **Toolbar:** Navigation bar ► SteeringWheels ► Basic View Object Wheel
Ribbon: Viewpoint tab ► Navigate panel ► Steering WheelsBasic

View Object 

 **Menu: Classic** user interface: View ► SteeringWheels ► View Object Wheel

 **Toolbar: Classic** user interface: Navigation Mode ► View Object Wheel 

Tour Building Wheels

With the Tour Building wheels (big and mini), you can move through a model, such as a building, an assembly line, ship, or oil rig. You can also walk through and navigate around a model. The big Tour Building wheel is optimized for new 3D users while the mini Tour Building wheel is optimized for experienced 3D users.



Big Tour Building Wheel

The big Tour Building wheel wedges have the following options:

- **Forward (page 248)**. Adjusts the distance between the current point of view and the defined pivot point of the model. Clicking once moves forward half the distance as far as the object you clicked.
- **Look**. Swivels the current view.

- **Rewind.** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Up/DownTool.** Slides the current view of a model along the Z axis of the model.

Mini Tour Building Wheel

The mini Tour Building wheel wedges have the following options:

- **Walk (Top wedge).** Simulates walking through a model.
- **Rewind (Right wedge).** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Up/Down (Bottom wedge).** Slides the current view of a model along the Z axis of the model.
- **Look (Left wedge).** Swivels the current view.

NOTE When the mini wheel is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

To switch to the mini Tour Building wheel

- Right-click the wheel, and click **Advanced Wheels ► Mini Tour Building Wheel.**

 **Toolbar:** Navigation bar ► SteeringWheels ► Mini Tour Building Wheel
Ribbon: Viewpoint tab ► Navigate panel ► Steering WheelsMini

Tour Building 

 **Menu:** Classic user interface: View ► SteeringWheels ► Mini Tour Building Wheel

 **Toolbar:** Classic user interface: Navigation Mode ► Mini Tour Building Wheel 

To switch to the big Tour Building wheel

- Right-click the wheel, and click **Basic Tour Building Wheel.**

 **Toolbar:** Navigation bar ► SteeringWheels ► Basic Tour Building Wheel
Ribbon: Viewpoint tab ► Navigate panel ► Steering WheelsBasic

Tour Building 

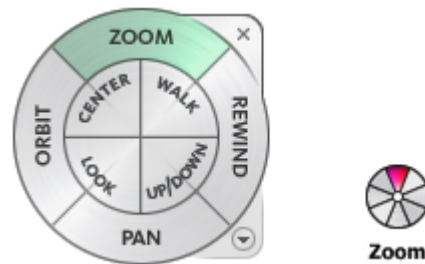
 **Menu:** **Classic** user interface: View ► SteeringWheels ► Tour Building Wheel

 **Toolbar:** **Classic** user interface: Navigation Mode ► Tour Building Wheel



Full Navigation Wheels

The Full Navigation wheels (big and mini) contain common 3D navigation tools used for both viewing an object and touring a building. The big and mini Full Navigation wheels are optimized for experienced 3D users.



NOTE When one of the Full Navigation wheels is displayed, you can press and hold the middle mouse button to pan, scroll the wheel button to zoom in and out, and hold the SHIFT key while pressing and holding the middle mouse button to orbit the model.

Big Full Navigation Wheel

The big Full Navigation wheel wedges have the following options:

- **Zoom.** Adjusts the magnification of the current view.
- **Rewind.** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Pan.** Repositions the current view by panning.
- **Orbit.** Rotates the current view around a fixed pivot point.
- **Center.** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- **Walk.** Simulates walking through a model.
- **Look.** Swivels the current view.

- **Up/Down.** Slides the current view of a model along the Z axis of the model.

Mini Full Navigation Wheel

The mini Full Navigation wheel wedges have the following options:

- **Zoom (Top wedge).** Adjusts the magnification of the current view.
- **Walk (Upper right wedge).** Simulates walking through a model.
- **Rewind (Right wedge).** Restores the most recent view. You can move backward or forward by clicking and dragging left or right.
- **Up/Down (Lower right wedge).** Slides the current view of a model along the Z axis of the model.
- **Pan (Bottom wedge).** Repositions the current view by panning.
- **Look (Lower left wedge).** Swivels the current view.
- **Orbit (Left wedge).** Rotates the current view around a fixed pivot point.
- **Center (Upper left wedge).** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.

To switch to the mini Full Navigation wheel

- Right-click the wheel, and click **Advanced Wheels ► Mini Full Navigation Wheel.**

 **Toolbar:** Navigation bar ► SteeringWheels ► Mini Full Navigation Wheel

Ribbon: Viewpoint tab ► Navigate panel ► Steering WheelsMini

Full Navigation 

 **Menu: Classic** user interface: View ► SteeringWheels ► Mini Full Navigation Wheel

 **Toolbar: Classic** user interface: Navigation Mode ► Mini Full Navigation Wheel 

To switch to the big Full Navigation wheel

- Right-click the wheel, and click **Full Navigation Wheel.**

 **Toolbar:** Navigation bar ► SteeringWheels ► Full Navigation Wheel

Ribbon: Viewpoint tab ► Navigate panel ► Steering WheelsFull

Navigation 

 **Menu: Classic** user interface: View ► SteeringWheels ► Full Navigation Wheel

 **Toolbar: Classic** user interface: Navigation Mode ► Full Navigation Wheel 

2D Navigation Wheel

With this wheel you can access basic 2D navigation tools; it is particularly useful when you do not have a pointing device with a scroll wheel.



The 2D Navigation wheel wedges have the following options:

- **Pan.** Repositions the current view by panning.
- **Zoom.** Adjusts the magnification of the current view.
- **Rewind.** Restores the most recent view orientation. You can move backward or forward by clicking and dragging left or right.

3Dconnexion 3D Mouse

A 3Dconnexion 3D mouse can be used as an alternative to the mouse to move around the **Scene View**.

The device has a pressure sensitive controller cap designed to flex in all directions. Push, pull, twist, or tilt the cap to pan, zoom, and rotate the current view. The speed of navigation is sensitive to the amount of force applied to the 3Dconnexion device. You can adjust the device settings by using the Control Panel for the device which is supplied by the manufacturer with the installation.



When a view change occurs with the 3Dconnexion 3D mouse, the ViewCube tool is reoriented to reflect the current view. You can change the behavior of the 3Dconnexion 3D mouse from the navigation bar.

3Dconnexion options on the navigation bar

Option	Description	Example
Object Mode	Navigates and reorients the view in the direction of the controller cap.	Move the controller cap right to pan the view to the right.
Walk Mode	Simulates walking through a model. The view of the model is moved in the opposite direction of the controller cap. The orientation and height of the current view is maintained.	Move the controller cap forward to walk towards the model.
Fly Mode	Simulates flying through a model. The view of the model is moved in the opposite direction of the controller cap. The orientation and height of the current view is not maintained.	Move the controller cap up to elevate the view. This makes the model appear to move down.

3Dconnexion options on the navigation bar		
Option	Description	Example
2D Mode	Navigates the view using only 2D navigation options. The view moves in the direction of the controller cap.	Move the controller cap to pan and zoom the view.
Center Tool	Specifies a point to define the pivot point and centers the model on that point. This pivot point is used by subsequent navigation tools until it is moved.	Click in the model. The view is centered on the model based on the specified point.
3Dconnexion Settings	Controls the default navigation mode and the speed of translation and rotation of the 3Dconnexion 3D mouse in the Options Editor (page 781)	Adjust the speed of translation and rotation.

If you are using the **Classic** user interface, the behavior of the 3Dconnexion device corresponds to the currently selected [navigation bar tool](#) (page 242) or [navigation mode](#) (page 264). This enables you to navigate with the 3Dconnexion device whilst performing other operations with the mouse. If no navigation tool or mode is selected or if the selected tool or mode is not a valid mode for the 3Dconnexion device, then a default navigation mode will be used.

Use View Management Keys in 3Dconnexion 3D Mouse

You can access different views (such as Top, Front, Left, Right, or Home) with buttons available on some 3Dconnexion 3D mouse models. Use the Button Configuration Editor to customize the operations of these buttons. When you click any of these buttons on the device, you can

- **Fit the view to the model extents.** Rotates the view of the object around the center of the scene and zooms out to fit the scene into the viewport.
- **Reorient the current view to a preset view.** Returns the view of the object to a predetermined view.

- **Maintain selection sensitivity.** Reorients the model around a defined pivot point based on the current selection.
- **Maintain lock to selection.** When Lock to Selection option is activated in the ViewCube tool, the view of the object reorients around the predefined center of the selected object.

Camera

Autodesk Navisworks offers you a number of prefixed options to control the camera projection, position, and orientation during navigation.

Set Camera Projection

You can choose to use a perspective camera or an orthographic camera during navigation in a 3D workspace only. In a 2D workspace an orthographic camera is always used.

NOTE Orthographic cameras are not available with **Walk** and **Fly** navigation tools.

To use a perspective camera

- Click **Viewpoint** tab ➤ **Camera** panel ➤ **Perspective**  .

 **Menu:** Classic user interface: **Viewpoint** ➤ **Navigation Tools** ➤ **Perspective**

To use an orthographic camera

- Click **Viewpoint** tab ➤ **Camera** panel ➤ **Orthographic**  .

 **Menu:** Classic user interface: **Viewpoint** ➤ **Navigation Tools** ➤ **Orthographic**

Control the Field of View

You can define the area of the scene that can be viewed through the camera in a 3D workspace only.

For the current viewpoint, you can move the FOV slider on the ribbon to adjust the Horizontal Field of View. For previously saved viewpoints, you can use the [Edit Viewpoint dialog box](#) (page 751) to adjust the values for both vertical and horizontal angles of view.

NOTE When you modify the Horizontal Field of View, the Vertical Field of View is automatically adjusted, and vice versa to match the aspect ratio in Autodesk Navisworks.

To control the horizontal field of view

- Click **Viewpoint** tab ➤ **Camera** panel, and move the FOV slider to control the camera's angle of view.
Moving the slider to the right produces a wider angle of view, and moving the slider to the left produces a narrower, or more tightly focused, angle of view.

Position and Focus Camera

You can adjust the camera's position and orientation in the scene.

Move Camera

For the current viewpoint, you can use the **Position** entry boxes on the ribbon to move the camera position. For previously saved viewpoints, you can use the [Edit Viewpoint dialog box](#) (page 751) to adjust the camera values.

NOTE The Z coordinate values are not available in a 2D workspace.

To move camera numerically

- 1 Click the **Viewpoint** tab, and slide out the **Camera** panel.
- 2 Type in numerical values into the **Position** entry boxes to move the camera by the amount entered.

Rotate Camera

You can adjust the angle of the camera during navigation in a 3D workspace only.

For the current viewpoint, use the **Tilt** window to rotate the camera up/down, and the **Roll** entry box on the ribbon to rotate the camera left/right. For saved viewpoints, you can use the [Edit Viewpoint dialog box](#) (page 751) to adjust the camera values.

Tilt Window



The tilt angle is indicated in the scene's units below (negative) or above (positive) horizontal (0) at the base of the window.

You can use the **Tilt** window with the **Walk** tool on the navigation bar to look up and down. If your mouse has a wheel, you can use it to adjust the tilt angle.

To toggle the Tilt window

- Click **Viewpoint** tab ► **Camera** panel ► **Show Tilt Bar** .

 **Command entry:** CTRL + F7

To roll camera up/down

- Drag the slider up or down on the **Tilt** window to roll the camera.

You can also type values directly the entry box at the base of the **Tilt** window. A positive value rotates the camera upwards, and a negative value rotates camera downwards. Typing 0 straightens the camera.

To roll camera left/right

- Click the **Viewpoint** tab, and slide out the **Camera** panel.
- Type in a value into the **Roll** entry box to rotate the camera around its front-to-back axis.
A positive value rotates the camera counterclockwise, and a negative value rotates it clockwise.

NOTE This value is not editable when the viewpoint up vector stays upright (that is, when you use **Walk**, **Orbit** and **Constrained Orbit** navigation tools).

Move Focal Point

You can change the focal point for the camera. For the current viewpoint, use the **Look At** entry boxes on the ribbon. For saved viewpoints, you can use the [Edit Viewpoint dialog box](#) (page 751) to adjust the camera values.

NOTE The Z coordinate values are not available in a 2D workspace.

In a 3D workspace, you can also put the **Scene View** into focus mode, which effectively swivels the camera so that the point clicked is in the center of the view. See [Focus](#) (page 310).

To move the camera focal point

- 1 Click the **Viewpoint** tab, and slide out the **Camera** panel.
- 2 Type in numerical values into the **Look At** entry boxes to move the camera focal point by the amount entered.

Straighten Camera

You can straighten the camera to align with the viewpoint up vector in a 3D workspace only.

When the camera position is close to the viewpoint up vector (within 13 degrees), you can use this function to snap the camera to the appropriate axis.

TIP The same effect can be achieved by typing 0 at the base of the **Tilt** window.

To straighten camera

- Click **Viewpoint** tab ► **Camera** panel ► **Align Camera** drop-down ► **Straighten** .

Predefined Camera Views

In Autodesk Navisworks, you can align a camera to one of the axis, or select one of six predefined face views to instantly change the camera's position and orientation in the scene. This functionality is available in a 3D workspace only.

When you align the camera position along one of the axis:

- Aligning with X axis toggles between front and back face views.
- Aligning with Y axis toggles between left and right face views.
- Aligning with Z axis toggles between top and bottom face views.

NOTE You can customize the location of the front face by using the ViewCube tool. This change is global, and affects all viewpoints.

To align with X-axis

- Click **Viewpoint** tab ► **Camera** panel ► **Align Camera** drop-down ► **Align X** .

 **Menu: Classic** user interface: Viewpoint ► Navigation Tools ► Align X

To align with Y-axis

- Click **Viewpoint** tab ► **Camera** panel ► **Align Camera** drop-down ► **Align Y** .

 **Menu: Classic** user interface: Viewpoint ► Navigation Tools ► Align Y

To align with Z-axis

- Click **Viewpoint** tab ► **Camera** panel ► **Align Camera** drop-down ► **Align Z** .

 **Menu:** **Classic** user interface: Viewpoint ► Navigation Tools ► Align Z

To look from a preset face view

- Right-click the **Scene View**, and click **Viewpoint ► Look From**.
- Click one of the face views. Choose from:
 - **Top**
 - **Bottom**
 - **Front**
 - **Back**
 - **Left**
 - **Right**

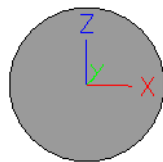
Navigation Aids

Head-Up Display

Head-up display elements are on-screen displays that provide information about your location and orientation in a 3D workspace. This functionality is not available in a 2D workspace.

In Autodesk Navisworks, you can use the following head-up display (HUD) elements:

- **XYZ Axes.** Shows the X, Y, Z orientation of the camera (or the avatar's eye if the avatar is visible). The **XYZ Axes** indicator is located at the bottom-left of the **Scene View**.



- **Position Readout.** Shows the absolute X, Y, Z position of the camera (or the avatar's eye position if the avatar is visible). The **Position Readout** is located at the bottom-left of the **Scene View**.

X: -50.51m Y: -57.92m Z: 10.97m

To toggle XYZ Axes

- 1 Click **View** tab ► **Navigation Aids** panel ► **HUD** drop-down.
- 2 Select or clear the **XYZ Axes** check box.

To toggle Position Readout

- 1 Click **View** tab ► **Navigation Aids** panel ► **HUD** drop-down.
- 2 Select or clear the **Position Readout** check box.

Reference Views

Reference views are useful to get an overall view of where you are in the whole scene and to quickly move the camera to a location in a large model. This functionality is available in a 3D workspace.

There are two types of reference views available in Autodesk Navisworks:

- Section View
- Plan View

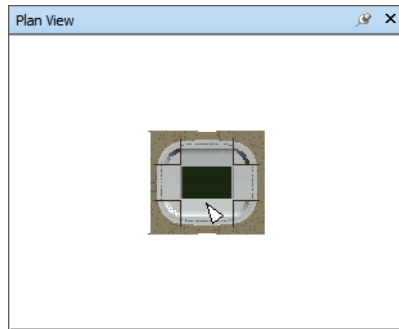
The reference views show a fixed view of the model. By default, the section view shows the view from the front of the model and the plan view shows a top view of the model.

Reference views are displayed inside the dockable windows. A triangular marker represents your current viewpoint. This marker moves as you navigate, showing the direction of your view. The marker may also be dragged by holding the left mouse button over it and dragging to move the camera in the **Scene View**.

NOTE The marker changes to a small dot when the reference view is in the same plane as the camera view.

To use the Plan View

- 1 Click **View** tab ► **Navigation Aids** panel ► **Reference Views** drop-down ► **Plan View** check box.
- The **Plan View** window opens with the reference view of the model.

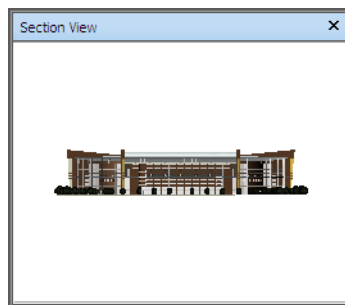


- 2 Drag the triangular marker on the reference view into a new location. The camera in the **Scene View** changes its position to match the position of the marker in the view.
Alternatively, navigate to a different location in the **Scene View**. The triangular marker in the reference view changes its position to match the camera position in the **Scene View**.
- 3 To manipulate a reference view, right-click anywhere in the **Plan View** window. Use the shortcut menu to adjust the view as desired.

 **Command entry:** CTRL + F9

To use the Section View

- 1 Click **View** tab ► **Navigation Aids** panel ► **Reference Views** drop-down ► **Section View** check box.
The **Section View** window opens with the reference view of the model.



- 2 Drag the triangular marker on the reference view into a new location. The camera in the **Scene View** changes its position to match the position of the marker in the view.

Alternatively, navigate to a different location in the **Scene View**. The triangular marker in the reference view changes its position to match the camera position in the **Scene View**.

- 3 To manipulate a reference view, right-click anywhere in the **Section View** window. Use the shortcut menu to adjust the view as desired.

 **Command entry:** CTRL + F10

Right-clicking the **Section View** or the **Plan View** window opens the shortcut menu with the following options.

Option	Description
Look From	Enables you to set the reference view to one of the preset viewpoints. Choose from: Top , Bottom , Front , Back , Left, Right , or Current Viewpoint . Selecting the Current Viewpoint option sets the reference view to the view in the active navigation viewpoint.
Update Current Viewpoint	Sets the active navigation viewpoint to the view in the reference view.
Edit Viewpoint	Opens the Edit Viewpoint dialog box, and enables you to modify the settings for the corresponding reference view.
Lock Aspect Ratio	Instructs Autodesk Navisworks to match aspect ratio of the reference view to that of the current viewpoint in the Scene View . The matching is done even when you resize the window with the reference view. This will usually give gray strips either to the top and bottom, or to either side of the reference view.
Refresh	Redraws the reference view based on the current setting. Reference view drawing uses software OpenGL and so can take a couple of seconds for large models.

Option	Description
Help	Opens context-sensitive help.

Focus

You can put the Scene View into focus mode until the next click.

When you are in focus mode, clicking on an item swivels the camera so that the point clicked is in the center of the view. This point becomes the focal point for the Orbit tools (SteeringWheels and navigation bar) in a 3D workspace only.

In the **Classic** user interface, 3D workspace, this point becomes the focal point for examine, orbit, and turntable [classic navigation modes](#) (page 264).

In a 2D workspace, the camera is moved to the center of the box of focused item while the z value remains the same.

To focus on an item

■ Click **Item Tools** tab ➤ **Look At** panel ➤ **Focus on Item** .

 **Toolbar:** Navigation bar ➤ Look tools ➤ Focus

 **Menu:** **Classic** user interface: Viewpoint ➤ Navigation Tools ➤ Focus

Shortcut menu: Scene ➤ Focus

Hold

When you navigate around a model in Autodesk Navisworks, it is possible to “pick up” or hold selected items and move around with them in the model.

For example you may be viewing a plan for a factory and would like to see different configurations of machine layouts.

To hold and release objects

- 1 Select the objects you want to hold either in the **Scene View** or in the Selection Tree.

- 2 Click **Item Tools** tab ➤ **Hold** panel ➤ **Hold** .

The selected objects are now held and will move with you through the model when you use navigation tools, such as **Walk**, **Pan** and so on.

- 3 To release the held objects, click **Hold**  on the ribbon again.
- 4 If you want to reset the objects to their original position, click **Item**

Tools tab ➤ **Transform** panel ➤ **Reset Transform** .

 **Menu: Classic** user interface: Viewpoint ➤ Navigation Tools ➤ Hold

Control the Realism of Your Navigation

When you navigate a 3D model, you can use the realism tools on the **Viewpoint** tab ➤ **Navigate** panel to control the speed and realism of your navigation. The realism tools are not available in a 2D workspace.

Gravity

NOTE This function only works in connection with collision.

Where collision gives you mass, gravity gives you weight. As such, you (as the collision volume) will be pulled downwards whilst walking through the scene.

NOTE Gravity can only be used with the Walk navigation tool.

This allows you to walk down stairs, for example, or follow terrain.

To toggle gravity

- When using the **Walk** tool, click **Viewpoint** tab ➤ **Navigate** panel ➤ **Realism** drop-down  ➤ **Gravity** check box.

 **Menu: Classic** user interface: Viewpoint ➤ Navigation Tools ➤ Gravity

 **Command entry:** CTRL + G

Crouching

NOTE This function only works in connection with collision.

When walking or flying around the model with collision activated, you may encounter object that are too low to walk under, a low pipe for example. This function enables you to crouch under any such objects.

With crouching activated, you will automatically crouch under any objects that you cannot walk under at your specified height, thereby not impeding your navigation around the model.

TIP To temporarily crouch under a low object, hold down the Space bar to allow navigation to proceed.

To toggle crouching

- 1 When using the **Walk** or **Fly** tool, click **Viewpoint** tab ► **Navigate** panel ► **Realism** drop-down  ► **Crouch** check box.

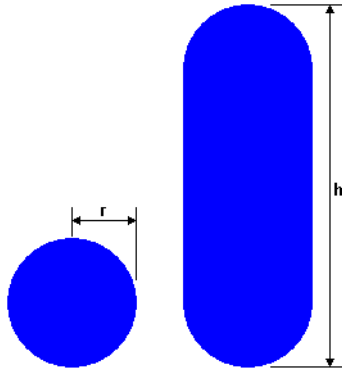
 **Menu: Classic** user interface: Viewpoint ► Navigation Tools ► Crouch

Collision

This function defines you as a collision volume - a 3D object that can navigate around and interact with the model, obeying certain physical rules that confine you within the model itself. In other words, you have a mass and as such, cannot pass through other objects, points or lines in the scene.

You can walk over, or climb over objects in the scene that are up to half the height of the collision volume, thus allowing you to walk up stairs, for example.

The collision volume, in its basic form, is a sphere (with radius = r), that can be extruded to give it height (with height = $h \geq 2r$). See diagram below:



The dimensions of the collision volume can be customized for the current viewpoint or as a global option.

NOTE Collision can only be used with the Walk and Fly navigation tools.

When collision is turned on, rendering prioritization is changed so that objects around the camera or avatar are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius and speed of movement (needing to see what is about to be walked into).

To toggle collision

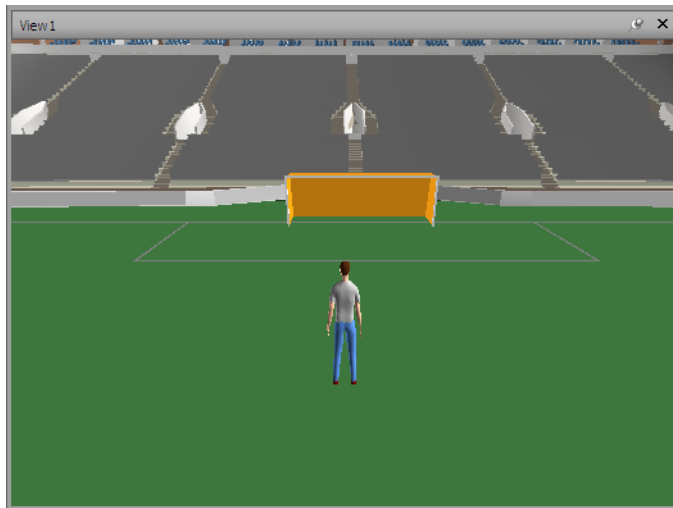
- 1 When using the **Walk** or **Fly** tool, click **Viewpoint** tab ➤ **Navigate** panel ➤ **Realism** drop-down  ➤ **Collision** check box.

 **Menu: Classic** user interface: Viewpoint ➤ Navigation Tools ➤ Collision
 **Command entry:** CTRL + D

Third Person View

This function allows you to navigate scene from a third person perspective.

When third person is activated, you will be able to see an avatar which is a representation of yourself within the 3D model. Whilst navigating you will be controlling the avatar's interaction with the current scene.



Using third person in connection with collision and gravity makes this a very powerful function, allowing you to visualize exactly how a person would interact with the intended design.

You can customize settings, such as avatar selection, dimension, and positioning, for the current viewpoint or as a global option.

When third person view is turned on, rendering prioritization is changed so that objects around the camera or avatar are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius, speed of movement (needing to see what is about to be walked into) and the distance of the camera behind the avatar (in order to see what the avatar is interacting with).

To toggle third person view

- 1 Click **Viewpoint** tab ► **Navigate** panel ► **Realism** drop-down ► **Third Person** check box. 

 **Menu:** Classic user interface: **Viewpoint** ► **Navigation Tools** ► **Third Person**

 **Command entry:** CTRL + T

To add and use a custom avatar

ATTENTION: Autodesk does not recommend or support usage of the custom avatars.

- 1 Open the file you want to use as your avatar (DWG, SKP, and so on) in Autodesk Navisworks.

- 2 Click the application button  ► **Save As**.

- 3 In the **Save As** dialog box, select .nwd in the **Save as Type** box.

- 4 Browse to the Autodesk Navisworks installation directory, for example:
C:\Program Files\Autodesk\Navisworks Manage
2012\avatars\my_new_folder_name.

- 5 Type in the new name for your avatar file, and click **Save**.

- 6 Restart Autodesk Navisworks, and open any file.

- 7 Click the application button  ► **Options**.

- 8 In the **Options Editor**, expand the Interface node, and click the **Viewpoint Defaults** option.

- 9 Click the **Settings** button in the **Collision** area.

- 10 In the **Default Collision** dialog box, select the **Enable** check box in the **Third Person** area.

- 11 Select your avatar in the **Avatar** drop-down list.

- 12 Click **OK** to return to the **Options Editor**.

- 13 You can also change the size of the avatar by changing the **Height** and the **Radius** values in the **Viewer** area.

- 14 Click **OK**.

- 15 Restart Autodesk Navisworks.

To change the default avatar

- 1 In Autodesk Navisworks, click the application button  ► **Options**.

- 2 In the **Options Editor**, expand the **Interface node**, and click the **Viewpoint Defaults** option.

- 3 Click the **Settings** button in the **Collision** area.

- 4 In the **Default Collision** dialog box, select the **Enable** check box in the **Third Person** area.

- 5 Click **OK** to return to the **Options Editor**.
- 6 Click **OK**.
- 7 Restart Autodesk Navisworks

To change an avatar for the current viewpoint

- 1 In Autodesk Navisworks, click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Edit Current Viewpoint**  on the ribbon.
- 2 In the **Edit Viewpoint** dialog box, click the **Settings** button in the **Collision** area.
- 3 In the **Collision** dialog box, select the **Enable** check box in the **Third Person** area.
- 4 Select a new avatar in the **Avatar** drop-down list.
- 5 Click **OK** to return to the **Edit Viewpoint** dialog box.
- 6 Click **OK**.

Control Model Appearance and Render Quality

6

Control Model Appearance

You can use the tools on the **Render Style** panel on the **Viewpoint** tab to control how your model is displayed in the **Scene View**.

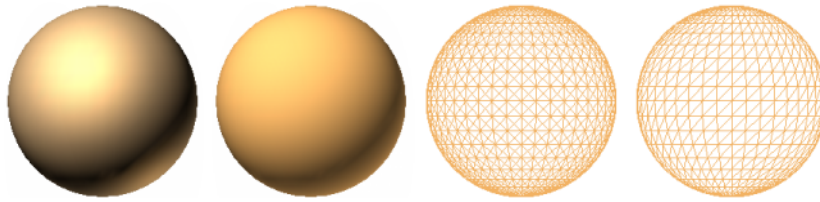
You have a choice of one of four interactive lighting modes (**Full Lights**, **Scene Lights**, **Head Light**, or **No Lights**), four rendering modes (**Full Render**, **Shaded**, **Wireframe**, or **Hidden Line**) and you can individually turn each of the five primitive types (**Surfaces**, **Lines**, **Points**, **Snap Points**, and **Text**) on and off.

NOTE Render and Lighting modes are not available in a 2D workspace.

Select Render Mode

Rendering shades the scene's geometry using the lighting you've set up, and the materials and environmental settings (such as background) you've applied.

In Autodesk Navisworks, you can use four render modes to control how the items are rendered in the **Scene View**. The spheres below demonstrate the effect that the render modes have on model appearance. In order from the left, these are **Full Render**, **Shaded**, **Wireframe**, and **Hidden Line**.



Full Render

In **Full Render** mode, the model is rendered with smooth shading including any materials that have been applied using the **Presenter** tool, or have been brought through from the native CAD file.

NOTE Autodesk Navisworks does not convert all native CAD file's textures. For more details, see [Use File Readers](#) (page 170) and [Use File Exporters](#) (page 192).

To select Full Render mode

- Click **Viewpoint** tab ► **Render Style** panel ► **Mode** drop-down, and click **Full Render** .

 **Menu:** Classic user interface: **Viewpoint** ► **Rendering** ► **Full Render**

Shaded

In **Shaded** mode, the model is rendered with smooth shading and without textures.

To select Shaded mode

- Click **Viewpoint** tab ► **Render Style** panel ► **Mode** drop-down, and click **Shaded** .

 **Menu:** Classic user interface: **Viewpoint** ► **Rendering** ► **Shaded**

Wireframe

In **Wireframe** mode, the model is rendered in wireframe. As Autodesk Navisworks uses triangles to represent surfaces and solids, all triangle edges are visible in this mode.

To select Wireframe mode

- Click **Viewpoint** tab ► **Render Style** panel ► **Mode** drop-down, and click **Wireframe** .

 **Menu:** Classic user interface: **Viewpoint** ► **Rendering** ► **Wireframe**

Hidden Line

In **Hidden Line** mode, the model is rendered in wireframe, but only the outline and facet edges of surfaces that are visible to the camera are displayed.

NOTE Unlike wireframe mode, where surfaces are rendered transparent, hidden line mode renders surfaces opaque.

To select Hidden Line mode

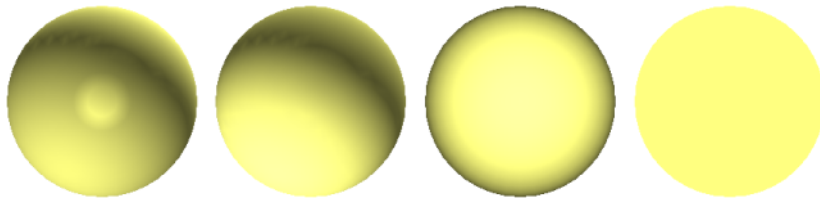
- Click **Viewpoint** tab ► **Render Style** panel ► **Mode** drop-down, and click **Hidden Line** .

 **Menu:** Classic user interface: **Viewpoint** ► **Rendering** ► **Hidden Line**

Add Lighting

In Autodesk Navisworks, you can use four lighting modes to control how the 3D scene is lit.

The spheres below demonstrate the effect the lighting styles have on them. In order from the left, these are **Full Lights**, **Scene Lights**, **Head Light**, and **No Lights**.



Full Lights

This mode uses lights that have been defined with the **Presenter** tool.

To use lights defined with the Presenter tool

- **Viewpoint** tab ► **Render Style** panel ► **Lighting** drop-down, and click **Full Lights** .

 **Menu:** Classic user interface: **Viewpoint** ► **Lighting** ► **Full Lights**

Scene Lights

This mode uses the lights that have been brought through from the native CAD file. If no lights are available, two default opposing lights are used instead.

You can customize the intensity of scene lights in the **File Options** dialog box.

To use lights defined with the model

- Click **Viewpoint** tab ► **Render Style** panel ► **Lighting** drop-down, and click **Scene Lights** .

 **Menu:** Classic user interface: **Viewpoint** ► **Lighting** ► **Scene Lights**

To adjust scene lights intensity

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, click the **Scene Lights** tab.
- 3 Move the **Ambient** slider to adjust the brightness of the scene.

TIP Turning on scene lights mode before following this procedure lets you instantly see the effect your changes have on the scene rendering.

- 4 Click **OK**.

Head Light

This mode uses a single directional light located at the camera that always points in the same direction as the camera.

You can customize the **Head Light** properties in the **File Options** dialog box (**Home** tab ► **Project** panel).

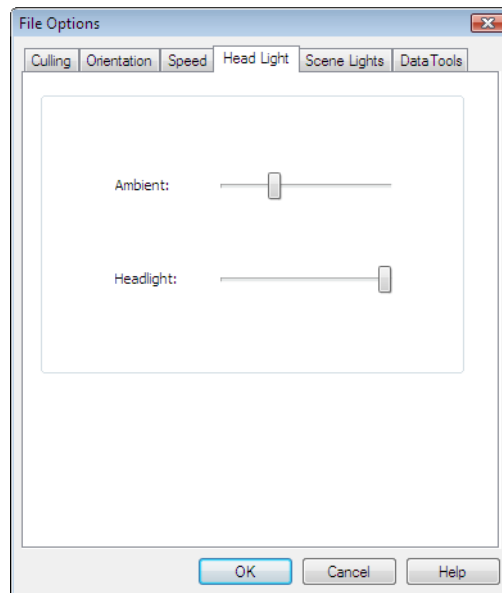
To use Head Light mode

- Click **Viewpoint** tab ► **Render Style** panel ► **Lighting** drop-down, and click **Head Light** .

 **Menu:** Classic user interface: **Viewpoint** ► **Lighting** ► **Head Light**

To adjust Head Light intensity

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, click the **Headlight** tab.



- 3 Move the **Ambient** slider to adjust the brightness of the scene, and the **Headlight** slider to adjust the brightness of the directional light.

TIP Turning on **Head Light** mode before following this procedure lets you instantly see the effect your changes have on the scene rendering.

- 4 Click **OK**.

No Lights

This mode switches off all lights. The scene is shaded with flat rendering.

To turn off all lights

- Click **Viewpoint** tab ► **Render Style** panel ► **Lighting** drop-down, and click **No Lights** .

 **Menu:** Classic user interface: **Viewpoint** ► **Lighting** ► **No Lights**

Select Background Effect

In Autodesk Navisworks, you can choose a background effect to use in the **Scene View**.

Currently, the following options are available:

- **Plain** - the background of the scene is filled with the selected color. This is the default background style. It can be used for 3D models and 2D sheets.



Plain background

- **Graduated** - the background of the scene is filled with a smooth gradient between the two selected colors. This background can be used for 3D models and 2D sheets.



Graduated background

- **Horizon** - the background of the 3D scene is split across the horizontal plane giving the effect of a sky and the ground. The resulting artificial horizon gives you an indication of your orientation in the 3D world. By default, the artificial horizon respects the world up vector as set in **File Options > Orientation**. This background is not supported for 2D sheets.

NOTE The artificial horizon is a background effect, and does not include a physical ground plane. So, for example, if you navigate “under the ground” and look up, you will not see the back of a ground plane, instead you will see the model from beneath, and a background filled with the sky color.



Horizon background

To set a plain background

- 1 Click **View** tab ► **Scene View** ► **Background** .
- 2 In the **Background Settings** dialog box, select **Plain** in the **Mode** drop-down list.
- 3 Select the required color from the **Color** palette.
- 4 Review the new background effect in the preview box, and click **OK**.

To set a graduated background

- 1 Click **View** tab ► **Scene View** ► **Background** .
- 2 In the **Background Settings** dialog box, select **Graduated** in the **Mode** drop-down list.
- 3 Select the first color from the **Top Color** palette.
- 4 Select the second color from the **Bottom Color** palette.
- 5 Review the new background effect in the preview box, and click **OK**.

To set an artificial horizon background for a 3D model

- 1 Click **View** tab ► **Scene View** ► **Background** .
- 2 In the **Background Settings** dialog box, select **Horizon** in the **Mode** drop-down list.
- 3 To set a graduated sky color, use the **Sky Color** and **Horizon Sky Color** palettes.
- 4 To set a graduated ground color, use the **Horizon Ground Color** and **Ground Color** palettes.
- 5 Review the new background effect in the preview box, and click **OK**.

Adjust Displaying of Primitives

You can enable and disable the drawing of **Surfaces**, **Lines**, **Points**, **Snap Points**, and 3D **Text** in the **Scene View**.

Points are “real” points in the model, whereas **Snap Points** mark locations on other primitives, for example the center of a circle, and are useful for snapping to when measuring.

Surfaces

Surfaces are the triangles that make up the 2D and 3D items in the scene. You can toggle the rendering of surfaces in the model.

To toggle the rendering of surfaces

- Click **Viewpoint** tab ► **Render Style** panel ► **Mode** drop-down, and click **Surfaces** .

 **Menu:** Classic user interface: **Viewpoint** ► **Display** ► **Surfaces**

Lines

You can toggle the rendering of lines in the model. You can also change the width of the drawn lines by using the **Options Editor**.

To toggle the rendering of lines

- Click **Viewpoint** tab ► **Render Style** panel ► **Lines** .

 **Menu:** Classic user interface: **Viewpoint** ► **Display** ► **Lines**

To change the line width

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** options.

- 3 On the **Display** page, **Primitives** area, enter a number between 1 and 9 in the **Line Size** box.
This sets the width in pixels for lines drawn in the **Scene View**.
- 4 Click **OK**.

Points

Points are real points in the model, for example, the points in a point cloud in a laser scan file. You can toggle the rendering of points in the model. You can also change the size of drawn points by using the **Options Editor**.

To toggle the rendering of points

- Click **Viewpoint** tab ➤ **Render Style** panel ➤ **Points** .

 **Menu:** Classic user interface: **Viewpoint** ➤ **Display** ➤ **Points**

To change the size of points

- 1 Click the application button  ➤ **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Primitives** area, enter a number between 1 and 9 in the **Point Size** box.
This sets the size in pixels for points drawn in the **Scene View**.
- 4 Click **OK**.

Snap Points

Snap points are implied points in the model, for example, the center point of a sphere or end points of a pipe. You can toggle the rendering of snap point in the 3D model. You can also change the size of the drawn snap points by using the **Options Editor**.

NOTE You cannot toggle the rendering of snap points for 2D sheets.

To toggle the rendering of snap points

- Click **Viewpoint** tab ► **Render Style** panel ► **Snap Points** .

To change the size of snap points

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Primitives** area, enter a number between 1 and 9 in the **Snap Size** box.
This sets the size in pixels of snap points drawn in the **Scene View**.
- 4 Click **OK**.

Text

You can toggle the rendering of text in 3D models. This functionality is not supported for 2D sheets.

To toggle the rendering of 3D text

- Click **Viewpoint** tab ► **Render Style** panel ► **Text** .

 **Menu:** Classic user interface: **Viewpoint** ► **Display** ► **Text**

Control Render Quality

Use Culling

Culling lets you navigate and manipulate large and complex scenes at interactive rates by intelligently hiding less-important objects as you work.

In Autodesk Navisworks, you can use the following methods of culling objects:

- **Area** - the objects' size in pixels determines whether the objects are rendered or not. By default, any objects smaller than 1x1 pixels in size are discarded.

- **Backface** - by default, only the front face of every polygon is drawn in Autodesk Navisworks. Sometimes, during the conversion process the front and back face of polygons get mixed, in which case, you need to adjust the Backface option.
- **Near and Far Clipping Planes** (frustum culling) - objects closer to the camera than the near clipping plane or beyond the far clipping plane are not drawn. You can let Autodesk Navisworks automatically constrain the location of the clipping planes, or you can constrain their location manually.

NOTE **Backface** and **Clipping Planes** are not used in a 2D workspace.

To set area culling

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **Culling** tab, select the **Enable** check box in the **Area** section.
- 3 Enter a value for the screen area in pixels below which geometry objects are culled. For example, setting this value to 100 pixels means that any object within the model that would be drawn less than 10x10 pixels in size are discarded.
- 4 Click **OK**.

To turn on backface culling for all objects in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **Culling** tab, select **On** in the **Backface** area.
- 3 Click **OK**.

To turn off backface culling for all objects in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **Culling** tab, select **Off** in the **Backface** area.
- 3 Click **OK**.

To turn on backface culling only for solid objects in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .

- 2 In the **File Options** dialog box, **Culling** tab, select **Solid** in the **Backface** area.
- 3 Click **OK**.

To constrain the position of the clipping planes automatically in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, click the **Culling** tab.
- 3 Select **Automatic** for the **Near** clipping plane.
- 4 Select **Automatic** for the **Far** clipping plane.
- 5 Click **OK**.

Autodesk Navisworks automatically controls the position of near and far clipping planes to give you the best view of the model.

To constrain the position of the clipping planes manually in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, click the **Culling** tab.
- 3 Select **Constrained** for the **Near** clipping plane, and enter the desired value in the **Distance** box.
- 4 Select **Constrained** for the **Far** clipping plane, and enter the desired value in the **Distance** box.
- 5 Click **OK**.

Autodesk Navisworks uses the provided values unless doing so affects the system performance (for example, makes the whole model invisible), in which case it adjusts the position of the clipping planes as necessary.

To fix the position of the clipping planes in a 3D workspace

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, click the **Culling** tab.
- 3 Select **Fixed** for the **Near** clipping plane, and enter the desired value in the **Distance** box.
- 4 Select **Fixed** for the **Far** clipping plane, and enter the desired value in the **Distance** box.
- 5 Click **OK**.

IMPORTANT Autodesk Navisworks uses the provided values even if doing so affects the system performance (for example, makes the whole model invisible).

Make Objects Required

Although Autodesk Navisworks intelligently prioritizes objects for culling in the scene, sometimes it drops out geometry that needs to remain visible while navigating.

You can make sure the objects are always rendered during interactive navigation by making them required.

To make objects required

- 1 Select geometry items that you want to remain visible during navigation in the **Selection Tree**.
 - 2 Click **Home** tab ► **Visibility** panel ► **Require** .
- In the **Selection Tree**, the object appear red when required.

TIP Clicking **Require**  again makes the selected objects unrequired.

 **Menu:** Classic user interface: **Edit** ► **Required**

 **Command entry:** CTRL + R

Ribbon: **Item Tools** tab ► **Visibility** panel ► **Require** 

To make all objects unrequired

- Click **Home** tab ► **Visibility** panel ► **Unhide All** drop-down ► **Unrequire All** .

Shortcut menu: **Scene** ► **Reset All** ► **Unrequire All**

Control Rendering of Objects

Adjust Scene Rendering During Navigation

Your models can range in size from small models to complex supermodels. As you navigate a scene in real time, Autodesk Navisworks automatically calculates which items to render first, based on the size of items, distance from the camera, and the specified frame rate. This customizable frame rate is guaranteed by default, but can be turned off, if necessary. Items that Autodesk Navisworks does not have time to render are dropped out. These dropped items are rendered when navigation stops.

The amount of drop-out depends on several factors, such as your hardware performance (graphics card and driver), the size of the **Scene View**, and the size of the model. When working with truly large supermodels in Autodesk Navisworks, you will require a sufficient amount of RAM to load and review the data.

Autodesk Navisworks employs JetStream technology which optimizes the usage of the available RAM. Before running out of memory, Autodesk Navisworks pages unnecessary data to the hard disk, freeing up space for loading to continue. JetStream technology also enables you to start navigating the supermodel, before it has been completely loaded into memory. Autodesk Navisworks is large address aware, and utilizes any additional memory assignment following the 3GB switch available on Windows XP systems.

TIP You can reduce the amount of drop-out during navigation by reducing frame rate, or switching off the **Guarantee Frame Rate** option.

To set the target frame rate

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **Speed** tab, select the number of frames per second to be applied to the rendered display of the model.
- 3 Click **OK**.

To set the level of detail

- 1 Click the application button  ► **Options**.

- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Detail** area, select the **Guarantee Frame Rate** check box to maintain the target frame rate during navigation. If this check box is clear, the complete model is rendered during navigation, no matter how long it takes.
- 4 Select the **Fill in Detail** check box to render a complete model when navigation stops. If this check box is clear, the items dropped out during navigation are not filled in when it stops.
- 5 Click **OK**.

To render transparent items

NOTE: If your video card supports hardware accelerated OpenGL, you can turn on the rendering of transparent items during interactive navigation. By default, transparent items are only drawn when interaction has ceased to prevent problems with display performance.

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Transparency** area, select the **Interactive Transparency** check box.
- 4 Click **OK**.

To render parametric primitives

NOTE: Modifying this option requires a restart of Autodesk Navisworks to take effect.

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Primitives** area, select the **Enable Parametric Primitives** check box. The level of detail changes during navigation depending on the distance from the camera.
If you want to use the default representations of primitives, clear this check box. The level of detail stays the same during navigation.
Click **OK**.

Accelerate Display Performance

If your video card supports OpenGL, you can improve the graphical performance by turning on hardware acceleration and occlusion culling.

Using the hardware acceleration usually gives you better and faster rendering. However, some graphics cards may not function well in this mode in which case switching this option off is recommended.

Occlusion culling can significantly improve performance in situations when much of the model is not visible. For example, when you walk down the corridor of a building, the walls occlude most geometry outside the corridor. Other rooms are only visible through doorways or windows. Turning on occlusion culling dramatically reduces the rendering load in such cases.

To use hardware acceleration

NOTE: If your video card does not support OpenGL hardware acceleration, this option is not available.

- 1 Click the application button  **> Options.**
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Acceleration** area, select the **Hardware Acceleration** check box. This allows Autodesk Navisworks to utilize any available OpenGL hardware acceleration on your video card.

NOTE If your video card drivers do not function well with Autodesk Navisworks, clear this check box.

- 4 Click **OK**.

To use occlusion culling

- 1 Click the application button  **> Options.**
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Display** option.
- 3 On the **Display** page, **Acceleration** area, select the **Occlusion Culling** check box.
- 4 Click **OK**.

NOTE Occlusion culling can only be used on a machine with an OpenGL 1.5 compliant graphics card. Also, occlusion culling is not used in a 2D workspace.

Adjust Presenter Materials

You can adjust the appearance of **Presenter** materials in the **Scene View** to get optimum performance from your graphics card when navigating around heavily textured scenes.

See also:

[Presenter Page](#) (page 824)

Stereo Rendering

Stereoscopic viewing in Autodesk Navisworks allows the viewing of the 3D model through stereo-enabled hardware, including active and passive stereo viewing glasses in conjunction with both CRT screens and dedicated projectors.

IMPORTANT Using stereo rendering requires the host computer have an OpenGL graphics card with stereo support. Additionally, some drivers require stereo to be explicitly enabled in the driver and may require lower color or resolution settings before the stereo rendering becomes available.

When the video output is in stereo mode, the view looks blurred without the correct glasses being worn. If the camera is in orthographic mode, it needs to be set to perspective for the effect to work correctly.

NOTE As the view for each eye has to be rendered separately it is not possible to support incremental filling in of detail in stereo mode. The detail appears when rendering is complete. Progress can be seen using the bar in the status area, and it is still possible to interrupt and start interacting again at any time.

In focal point-based navigation modes (such as orbit, free orbit and constrained orbit) the model is positioned so that parts closer than the focal point appear in front of the screen, with the rest behind. In other navigation modes (such as walk and fly), the focal point is set so that any avatar will be level with the screen. Objects between you and the avatar will appear in front of the screen. You can adjust the out of screen effect in the **Stereo Options** dialog box.

To enable stereo rendering

- Click **View** tab ► **Stereo** panel ► **Enable Stereo**  .

NOTE This option is only available if you have the required hardware, and the correct driver and display settings.

To adjust the stereo effects

- 1 Click **View** tab ► **Stereo** panel ► **Stereo Options** tool launcher  .
- 2 In the **Stereo Options** dialog box, move the **Magnitude** slider to vary the strength of the effect.
- 3 To adjust the out of screen effect, select the **Enable** check box, and then move the slider to control how much of the scene appears out of the screen during navigation.
- 4 If you need to swap the left and right eyes over, select the **Swap Eyes** check box. This can be useful when moving from CRT to a large screen projector and back.
- 5 Click **OK**.

Review Your Model

7

Select Objects

With large models it is potentially a very time-consuming process to select items of interest. Autodesk Navisworks makes this a much simpler task by providing a range of functions for quickly selecting geometry both interactively and by searching the model manually and automatically.

Interactive Geometry Selection

In Autodesk Navisworks, there is a concept of an active selection set (the currently selected items, or the current selection) and saved selections sets. Selecting and finding items makes them part of the current selection, so you can hide them or override their colors. At any time, the current selection can be saved and named for retrieval in later sessions.

Selecting items makes them part of the current selection, so you can hide them or override their colors.

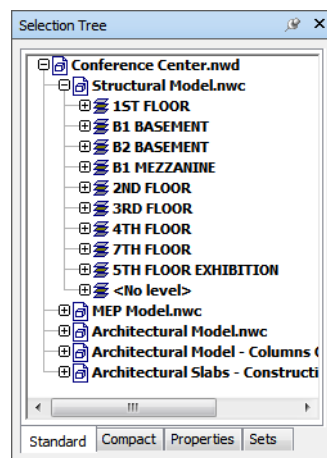
You can use several methods to interactively select items into the current selection. You can use the tabs in the **Selection Tree**, select items directly in the **Scene View** with the **Select** and **Select Box** tools, and you can select other items with similar properties to an existing selection using the selection commands.

NOTE Right-clicking any item in the **Selection Tree** or **Scene View** opens a shortcut menu.

You can also customize the level at which you select items ([selection resolution](#)) (page 345), and modify the highlighting method for the items selected in the **Scene View**.

Selection Tree Window

The **Selection Tree** is a dockable window, which displays a variety of hierarchical views of the structure of the model, as defined by the CAD application in which the model was created.



Autodesk Navisworks uses this hierarchical structure to identify object-specific paths (from the file name down to a particular object).

By default there are four tabs:

- **Standard.** Displays the default tree hierarchy, including all instancing. The contents of this tab can be sorted alphabetically.
- **Compact.** Displays a simplified version of the hierarchy on the **Standard** tab, omitting various items. You can customize the level of complexity of this tree in the **Options Editor**.
- **Properties.** Displays the hierarchy based on the items' properties. This enables simple manual searching of the model by item property.
- **Sets.** Displays a list of selection and search sets. If no selection and search sets have been created, this tab is not shown.

NOTE The list of the items on the **Sets** tab is exactly the same as the list on the **Sets** dockable window.

Additional customized **Selection Tree** tabs can be added by using the Autodesk Navisworks API.

Naming of items reflects the names from the original CAD application, wherever possible. You can copy and paste names from the **Selection Tree**. To do this, right-click an item in the **Selection Tree**, and click **Copy Name** on the context menu. Alternatively, you can click an item in the **Selection Tree**, and press CTRL + C. The name is now copied to the clipboard.

There are different tree icons representing the types of geometry making up the structure of the model. Each of these item types can be marked as hidden (gray), unhidden (dark blue) or required (red).

NOTE If a group is marked as hidden or required, then all instances of that group are marked as hidden or required. If you want to operate on a single occurrence of an item, then you should mark the instanced group (the level above, or the “parent”, in the hierarchy) hidden or required.

To toggle the Selection Tree

■ Click **Home** tab ► **Select & Search** panel ► **Selection Tree** .

 **Menu:** Classic user interface: **View** ► **Control Bars** ► **Selection Tree**

 **Command entry:** CTRL + F12

To use the Selection Tree to select objects

- 1 Open the **Selection Tree**, and click the **Standard** tab.
- 2 Click an object in the **Selection** tree to select the corresponding geometry in the **Scene View**.

NOTE When you select an item in the tree, individual geometry or a group of geometry is selected in the **Scene View** depending on chosen selection resolution.

- 3 To select several items at the same time, use the SHIFT and CTRL keys. CTRL allows multiple selection item by item, and SHIFT allows multiple selection between the first and last items selected.
- 4 To remove selection from an object in the **Selection Tree** press ESC.

To change the sort order on the Standard tab

- 1 Open the **Selection Tree**, and click the **Standard** tab.

- 2 Right-click any item in the tree and click **Scene ► Sort**. The contents of the tab is now ordered alphabetically.

NOTE You cannot use the **Undo** ↶ option on the **Quick Access** toolbar to reverse this action.

To customize the contents of the Compact tab

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Selection** option.
- 3 On the **Selection** page, select the required level of detail in the **Compact Tree** box. Choose from the following options:
 - **Models** - the tree is restricted to displaying model files only.
 - **Layers** - the tree can be expanded down to the layer level.
 - **Objects** - can be expanded down to the objects level, but without the levels of instancing shown on the **Standard** tab.
- 4 Click **OK**.

Icon	Description
	A model, such as a drawing file or design file.
	A layer or level.
	A group, such as a block definition from AutoCAD or cell definition from MicroStation.
	An instanced group, such as an inserted block from AutoCAD or cell from MicroStation. If in the imported file the instance was unnamed, Autodesk Navisworks names the instance to match its child's name.
	An item of geometry, such as a polygon.

Icon	Description
	An instanced item of geometry, such as an instance from 3D Studio.
	A composite object. A single CAD object that is represented in Autodesk Navisworks by a group of geometry items.
	Saved selection set.
	Saved search set.

Selection Tools

There are two selection tools (Select  and Select Box ) available from **Home** tab ► **Select & Search** panel to control the way you select geometry.

Typically, using selection tools is mutually exclusive to using navigation tools (see [Product-Specific Navigation Tools](#) (page 241), so that when you are selecting you cannot navigate and vice versa.

NOTE When using a 3Dconnexion 3D mouse in conjunction with the standard mouse control, the 3Dconnexion device can be configured for navigation and the mouse for selecting. See [3Dconnexion 3D Mouse](#) (page 298) for more information.

Selecting geometry in the **Scene View** automatically selects the corresponding objects in the **Selection Tree**.

Holding the SHIFT key whilst selecting items in the **Scene View** cycles through the selection resolution, allowing you to get more specific with your selections.

You can use the **Options Editor** to customize the distance from an item you have to be for it to be selected (pick radius). This is useful when you select lines and points.

Select Tool

The Select tool lets you select items in the **Scene View** with a mouse click. The tool is activated by clicking **Home** tab ► **Select & Search** panel

► **Select** drop-down ► **Select**  . Once a single item is selected, its properties are shown in the **Properties** window.

Select Box Tool

In select box mode, you can select multiple items in the model by dragging a rectangular box around the area you want to make your current selection.

To select geometry with the Select tool

- 1 Click **Home** tab ► **Select & Search** panel ► **Select** drop-down ► **Select**  .
- 2 Click an item in the **Scene View** to select it.
- 3 To select multiple geometry, press and hold down the CTRL key while clicking items in the scene.
- 4 To remove items from the current selection, hold down the CTRL key while clicking them again. Alternatively, press the ESC key to remove all items from the current selection.

 **Menu:** Classic user interface: **Edit** ► **Select** ► **Select**

 **Command entry:** CTRL + 1

To select geometry with the Select Box tool

- 1 Click **Home** tab ► **Select & Search** panel ► **Select** drop-down ► **Select Box**  .
- 2 Drag a box with the left mouse button over the **Scene View** to select all items within the box.

TIP Holding down the SHIFT key while dragging the box selects all items within and that intersect the box.
- 3 To select multiple geometry, press and hold down the CTRL key while dragging a box in the scene.
- 4 To remove items from the current selection, press the ESC key.

To set the pick radius

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the Interface node, and click the **Selection** option.
- 3 On the **Selection** page, enter the radius in pixels that an item has to be within in order to be selected. The valid values are between 1 and 9.
- 4 Click **OK**.

Selection Commands

Selection commands enable you to quickly alter the current selection using logic. You can select multiple items based on the currently selected items' properties, or quickly invert the set, select everything or nothing.

To select all items within the model

- Click **Home** tab ► **Select & Search** panel ► **Select All** drop-down ► **Select All**  .

To deselect all items

- Click **Home** tab ► **Select & Search** panel ► **Select All** drop-down ► **Select None**  .

To invert your current selection

- Click **Home** tab ► **Select & Search** panel ► **Select All** drop-down ► **Invert Selection**  .

Currently selected items become deselected, and currently deselected items become selected.

To select all instances of the selected geometry group

- Click **Home** tab ► **Select & Search** panel ► **Select Same** drop-down ► **Select Multiple Instances**  .

To select all items with the same name as the currently selected item

- 1 Click **Home** tab ► **Select & Search** panel ► **Select Same** drop-down ► **Same Name** .

To select all items with the same type as the currently selected item

- Click **Home** tab ► **Select & Search** panel ► **Select Same** drop-down ► **Same Type** .

To select all items with the same property as the currently selected item

- Click **Home** tab ► **Select & Search** panel ► **Select Same** drop-down ► **Same** <Property>.

To use a saved selection or search set

- Click **Home** tab ► **Select & Search** panel ► **Sets** drop-down, and click the set you want to recall.

The selection commands are as follows:

- **Select All.** Selects all items contained within the model.
- **Select None.** Deselects everything in the model.
- **Invert Selection.** Currently selected items become deselected and vice versa.
- **Sets.** Provides you with options to save and recall selection and search sets.
- **Select Multiple Instances.** Selects all instances (sometimes called insertions) of the currently selected geometry group that occur in the model.
- **Select Same Name.** Selects all items in the model that have the same name as the currently selected item.
- **Select Same Type.** Selects all items in the model that have the same type as the currently selected item.
- **Select Same** <Property>. Selects all items with the same property as the currently selected item. This property can be any searchable property currently attached to the item, for example material or link.

NOTE Using the **Select Same** <Property> command works by comparing items' properties. If you have multiple items selected when you perform a selection command of same name or type and so on, all the types, names and properties of the items in the current selection are compared with all items' properties in the scene. The items with properties matching any properties of the currently selected items are selected.

Set Selection Resolution

When you click an item in the **Scene View**, Autodesk Navisworks doesn't know what level of item to start selecting at - do you mean the whole model, or the layer, or the instance, or group, or just the geometry? The default selection resolution specifies a starting point for the object path in the **Selection Tree** so that Autodesk Navisworks can locate and select the item.

You can customize the default selection resolution on the **Home** tab ► **Select & Search** panel. Alternatively, you can use the **Options Editor**. Or you can use a quicker way, by right-clicking any item in the **Selection Tree** and clicking **Set Selection Resolution to X**, where "X" is one of the available selection resolutions.

If you find you have selected the wrong level of item, you can interactively cycle through the selection resolution, without having to go to the **Options Editor** or the **Home** tab. You can do this by holding down the SHIFT key when clicking an item. This selects an item one level more specific each time you click the item until the resolution gets to "geometry", at which point it reverts back to "model". Clicking on a different item reverts the selection resolution back to default (as set in the **Options Editor**).

To change the default selection resolution in the Options Editor

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Selection** option.
- 3 On the **Selection** page, select the required starting point for the object path in the **Resolution** box.
- 4 Click **OK**.

To change the default selection resolution with the ribbon tools

- 1 Click the **Home** tab and expand the **Select & Search** panel.
- 2 Click the **Selection Resolution** drop-down, and select the required option.

The available options for selection resolution are as follows:

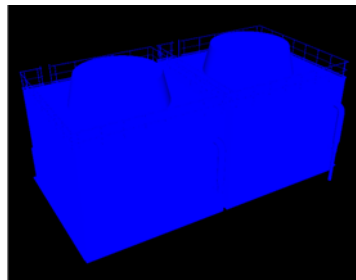
- **File.** Makes the object path start at the file level; as a result, all objects at the current file level are selected.
- **Layer.** Makes the object path start at the layer node; as a result all objects within a layer are selected.
- **First Object.** Makes the object path start at the highest level of objects below the layer node, if applicable.
- **Last Object.** Makes the object path start at the lowest level of objects in the **Selection Tree**. Autodesk Navisworks looks for composite objects first, and if none are found, the geometry level is used instead. This is the default option.
- **Last Unique.** Makes the object path start at the first unique level of objects (not multiple-instanced) in the **Selection Tree**.
- **Geometry.** Makes the object path start from the geometry level in the **Selection Tree**.

Set Highlighting Method

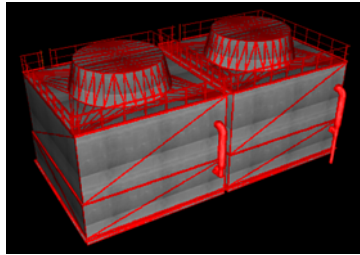
You can use the **Options Editor** to customize color and method of highlighting geometry selected in the **Scene View**.

There are three types of highlighting:

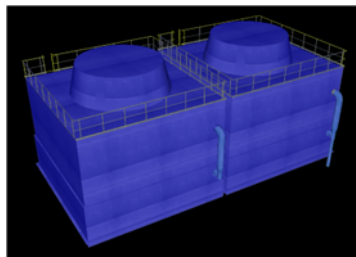
- Shaded



- Wireframe



■ Tinted



To toggle highlighting of selected objects

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Selection** option.
- 3 On the **Selection** page, **Highlight** area, select the **Enabled** check box, if you want the selected items to be highlighted in the **Scene View**. Clear this check box, if you don't want any highlighting.
- 4 Click **OK**.

To customize the way objects are highlighted

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Selection** option.
- 3 Make sure the **Enabled** check box is selected.
- 4 Use the **Method** drop-down list to select the type of highlighting you want (**Shaded**, **Wireframe** or **Tinted**).

- 5 Click the **Color** palette to select the highlight color.
- 6 If you selected **Tinted** in the **Method** box, use the slider to adjust the **Tint Level**.
- 7 Click **OK**.

Hide Objects

Autodesk Navisworks provides tools that can be used to hide and display objects or groups of objects. Hidden objects are not drawn in the **Scene View**.

Hide Selected Objects

You can hide the objects in the current selection so that they are not drawn in the **Scene View**. This is useful when you want to remove specific parts of the model. For example, when you walk down the corridor of building, you may want to hide a wall that occlude your view of the next room.

Hide Unselected Objects

You can hide all items except those currently selected so that they are not drawn in the **Scene View**. This is useful when you only want to see specific parts of the model.

NOTE In the **Selection Tree**, the items appear gray when marked as hidden.

To hide selected objects

- 1 In the **Scene View**, select all items you want to hide.
- 2 Click **Home** tab ► **Visibility** panel ► **Hide** .
The selected objects are now invisible.

TIP Clicking **Hide**  again displays the invisible objects.

 **Menu:** Classic user interface: **Edit** ► **Hidden**

 **Command entry:** CTRL + H

Shortcut menu: **Hide**

To make unselected items hidden

- 1 In the **Scene View**, select all items you want to review.
- 2 Click **Home** tab ► **Visibility** panel ► **Hide Unselected** . Only the selected geometry remains visible.

TIP Clicking **Hide Unselected**  again displays the invisible objects.

 **Menu:** Classic user interface: **Edit** ► **Hide Unselected**

Shortcut menu: Hide Unselected

To reveal all hidden objects

- Click **Home** tab ► **Visibility** panel ► **Unhide All** drop-down ► **Unhide All** .

Shortcut menu: **Scene** ► **Reset All** ► **Unhide All**

Find Objects

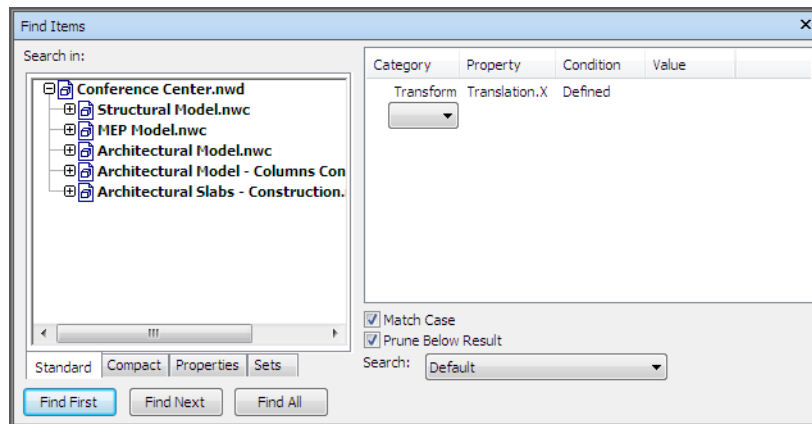
Finding is a quick and powerful way of selecting items into the current selection based on items' properties.

You can use the **Find Items** window to set up and run a search, which can then be saved and re-run in later sessions or shared with other users.

You can also use **Quick Find**, which is a faster way of searching. It simply looks for the specified string in all property names and values attached to items in the scene.

Find Items Window

The **Find Items** window is a dockable window that enables you to search for items having a common property or combination of properties.



The left pane contains the **Find Selection Tree** with several tabs at the bottom, and enables you to select the level of item to start searching at; this can be a file, a layer, an instance, a selection set, and so on.

The tabs are the same as on the **Selection Tree** window:

- **Standard.** Displays the default tree hierarchy, including all instancing.
- **Compact.** Displays a simplified version of the tree hierarchy.
- **Properties.** Displays the hierarchy based on the items' properties.
- **Sets.** Displays a list of selection and search sets. If no selection and search sets have been created, this tab is not shown.

NOTE The list of the items on the Sets tab is exactly the same as the list on the **Sets** window.

The right pane enables you to add the search statements (or conditions). And the buttons enable you to find the qualifying items in your scene.

Defining Search Statements

A search statement contains a property (a combination of category name and property name), a condition operator, and a value to be tested against the selected property. For example, you can search for **Material** that **ContainsChrome**.

By default, all items that match your statement criteria are found (for example, all objects that use chrome material). You can also negate a statement, in which case, all items that do not match your statement criteria are found instead (for example, all objects that do not use chrome material).

Each category and property name has two parts - a user string which is shown in the Autodesk Navisworks interface, and an internal string which is not shown, and is mainly used by the API. By default, the items are matched on both parts, but you can instruct Autodesk Navisworks to match only on one part, if necessary. For example, you can ignore user names in searches, and match items on their internal names only. This can be useful when you plan on sharing your saved searches with other users who may be running localized versions of Autodesk Navisworks.

The statements that do not use default settings are identified by this icon: ✱. It appears, for example, when you negate a statement, or choose to ignore the upper and lower cases in property values.

Combining Search Statements

Search statements are read from left to right. By default, all statements are ANDed. For example: “A AND B”, “A AND B AND C”.

You can arrange your statements into groups. For example, “(A AND B) OR (C AND D)”. ORed statements are identified by a plus icon: +. All statements preceding the ORed statement are ANDed, and all statements following the ORed statement are ANDed. So to create two groups in the previous example, you need to mark statement C as ORed.

There are no parentheses to visually show you the way the statements are read. Simple statements such as “A OR B” will not be misinterpreted. For complex searches, the order and groupings of the statements is more important, especially if you choose to negate some statements. For example, “(A AND B) OR (C AND NOT D)”. When search conditions are evaluated, NOT is applied before AND, and AND is applied before OR.

To toggle the Find Items window

■ Click **Home** tab ► **Select & Search** panel ► **Find Items** .

 **Menu:** Classic user interface: **View** ► **Control Bars** ► **Find Items**

 **Command entry:** SHIFT + F3

To find objects

- 1 Open the **Find Items** window.
- 2 On the **Find Selection Tree**, click the items where you want to start searching from. For example, if you want to search the whole model, click the **Standard** tab, press and hold CTRL, and click all files that

comprise the model. If you want to limit your search to a selection set, click the **Sets** tab, and click the required set.

- 3 Define a search statement:
 - a Click the **Category** column, and select the property category name from the drop-down list, for example, 'Item'.
 - b In the **Property** column, select the property name from the drop-down list, for example, 'Material'.
 - c In the **Condition** column, select the condition operator, for example, "Contains".
 - d In the **Value** column, type in the property value to search for, for example, "Chrome".
 - e If you want to make your search statement case-insensitive, right-click it, and click **Ignore String Value Case**.
- 4 Define more search statements, if required.

By default, all statements are ANDed. This means that they all need to be true for an item to be selected. You can make a statement use OR logic, by right-clicking it and clicking **Or Condition**. If you are using two statements, and marked the second one as ORed, this means that an item will be selected if either of these statements is true.
- 5 Click the **Find All** button. The search results are highlighted in the **Scene View** and the **Selection Tree**.

To save current search

- 1 Click **Home** tab ► **Select & Search** panel ► **Sets** drop-down ► **Manage Sets**.

This opens the **Sets** window and makes it the active window.
- 2 Right-click anywhere in the **Sets** window, and click **Add Current Search**.
- 3 Type a name for your search set, and press ENTER.

 **Menu:** Classic user interface: **Edit** ► **Select** ► **Selection Sets** ► **Add Current Search**

To export current search

- 1 Click **Output** tab ► **Export Data** panel ► **Current Search** .
- 2 In the **Export** dialog box, browse to the desired folder.

- 3 Enter a name for your file, and click **Save**.

To import a saved search

- 1 Click the application button  > **Import** > **Search XML**.
- 2 In the **Import** dialog box, browse to the folder containing the file with the save search criteria, and select it.
- 3 Click **Open**.

Search Options

Category Selects the category name. Only the categories that are contained in the scene are available in the drop-down list.

Property Selects the property name. Only the properties in the scene within the category chosen are available in the drop-down list.

Condition Selects a condition operator for your search. You can use the following operators depending on the property you are searching for:

- **=**. Equals; can be used to evaluate any type of property. To qualify for the search, a property must match the specified value exactly.
- **Not Equals**. Can be used to evaluate any type of property.
- **> -**. Greater Than; can be used to evaluate numerical property types only.
- **>=**. Greater Than or Equals; can be used to evaluate numerical property types only.
- **< -**. Less Than; can be used to evaluate numerical property types only.
- **<=**. Less Than or Equals; can be used to evaluate numerical property types only.
- **Contains**. To qualify for the search, a property must contain the specified value (for example, series of letters within a string).
- **Wildcard**. Enables you to use wildcards in the **Value** field to allow matching against any character (?) or an arbitrary sequence of characters (*).
- **Defined**. To qualify for the search, a property must have some value defined.
- **Undefined**. To qualify for the search, a property must not have any defined value.

Value You can either type in a value freely in this box, or choose a pre-defined value from the drop-down list which shows all values in the scene available within the category and property you defined earlier. If you used **Wildcard** as the condition operator, you can type a value with wildcards. To match one single unspecified character use the symbol “?” (question mark). To match any number of unspecified characters, use the symbol * (asterisk). For example, “b??k” will match “brick” and “block”; “b*k” will match “bench kiosk”, “brick”, and “block”; and “*b*k*” will match “bench kiosk”, “brick” and “block” and also “Coarse bricks” and “block 2”.

This box is not available if you used **Defined** or **Undefined** as the condition operator.

Match Case Select this check box to respect the upper and lower case letters of the values tested during the search. This affects all statements in your search. It's also possible to set case sensitivity for individual search statements by using the Search Conditions shortcut menu.

Prune Below Result Select this check box if you want to stop searching a branch of the **Find Selection Tree** as soon as the first qualifying object is found.

Search Specifies the type of the search to run. Choose from:

- **Default.** Searches all items selected in the **Find Selection Tree**, along with the paths below these items, for qualifying objects.
- **Below Selected Paths.** Only searches below the items selected in the **Find Selection Tree** for qualifying objects.
- **Selected Paths Only.** Only searches within the items selected in the **Find Selection Tree** for qualifying items.

Find Selection Tree Shortcut Menu

Select Changes the selection you made in the **Scene View** to the current selection in the **Find Selection Tree**.

Import Current Selection Changes the selection in the **Find Selection Tree** to your current selection the **Scene View**.

Search Conditions Shortcut Menu

Ignore String Value Case Makes the selected search statement case-insensitive (for example, both ‘Chrome’ and ‘chrome’ materials are treated as qualifying).

Ignore Category User Name Instructs Autodesk Navisworks to use internal category names and ignore the user category names for the selected search statement.

Ignore Category Internal Name Instructs Autodesk Navisworks to use user category names and ignore the internal category names for the selected search statement.

Ignore Property User Name Instructs Autodesk Navisworks to use internal property names and ignore the user property names for the selected search statement.

Ignore Property Internal Name Instructs Autodesk Navisworks to use user property names and ignore the internal category names for the selected search statement.

Or Condition Chooses OR condition for the selected search statement.

Negate Condition Negates the selected search statement, so that all items that do not match the statement criteria are found.

Delete Condition Deletes the selected search statement.

Delete All Conditions Deletes all search statements.

Buttons

Find First Finds the first qualifying item, and selects it in the **Scene View** and the **Selection Tree**.

Find Next Finds the next qualifying item, and selects it in the **Scene View** and the **Selection Tree**.

Find All Finds all qualifying items, and highlights them in the **Scene View** and the **Selection Tree**.

TIP Click **Home** tab ► **Visibility** ► **Hide Unselected**  to display only the search results in the **Scene View**.

Quick Find

To locate and select the objects quickly, use the **Quick Find** feature.

To quickly find items

- 1 Click **Home** tab ► **Select & Search** panel.

- 2 In the **Quick Find** text box, type in the string to search for in all item's properties. This can be a word or a few words. The search is not case-sensitive.
- 3 Click **Quick Find** . Autodesk Navisworks finds and selects the first item in the **Selection Tree** that matches the entered text, selects it in the **Scene View**, and stops the search.
- 4 To find more items, click **Quick Find**  again. If there are any more items that match the entered text, Autodesk Navisworks selects the next one in the **Selection Tree**, selects it in the **Scene View**, and stops the search. Subsequent clicks find next instances.

 **Command entry:** To open the **Quick Find** dialog box: CTRL + F. To Find Next: F3

Find All Sheets and Models Containing the Selected Object

You can search for an object across all prepared sheets/models in the **Project Browser**.

Models exported into Autodesk Navisworks can be integrated with 2D sheets exported from the same design application to form a multi-sheet file. For example models exported from Autodesk Revit to the NWC, DWF or DWF(x) file format can be integrated with any 2D sheets exported to DWF/DWF(x) files from the same Revit project. This allows you to select a model component in the 3D environment, and then to find and review the same component in a 2D representation (such as a floor plan or section). Similarly, you can select an item in your 2D sheet and find it in the 3D model and other 2D sheets. This can enable you, for example, to quickly review a clashing item on a floor plan of the relevant system.

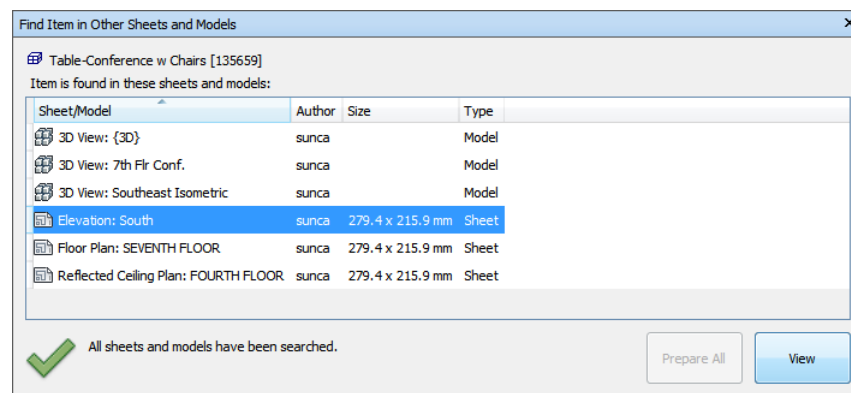
Preparing Sheets/Models for Searching

When you open a multi-sheet file, not all sheets/models may have been prepared to be used in Autodesk Navisworks. All sheets/models which require preparation are indicated with the Prepare  icon in the **Project Browser** window. You can use the **Project Browser** shortcut menu to prepare one or more sheets/models (see [Project Browser Window](#) (page 214)), or you can prepare all sheets/models directly in the **Find Items in Other Sheets and Models** window. Autodesk Navisworks only searches prepared sheets/models.

So if some sheets/models in your file have not been prepared, they will not be included in the search.

Find Items in Other Sheets and Models Window

The **Find Items in Other Sheets and Models** is a dockable window that enables you to search for an object across all prepared sheets/models in the **Project Browser**.



The label at the top of the window indicates the object currently selected in the **Scene View**. The geometry type is represented with an [icon](#) (page 340) next to the object's name. The icons are the same as the icons used on the **Selection Tree** window. The search results are displayed in the Sheets/Models list, and contain all sheets/models in the currently open file where the selected object has been found. You can sort the results by clicking the heading of the desired column. This alternates the sort order between ascending and descending.

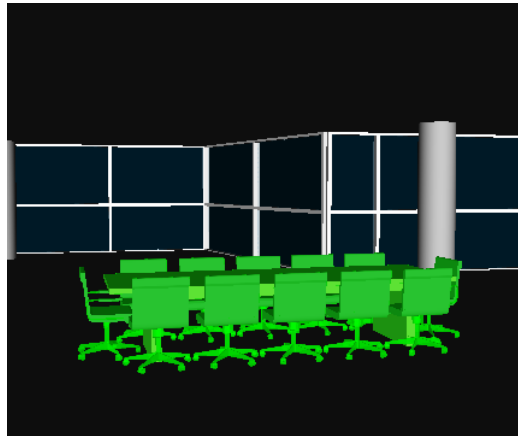
To toggle the Find Item in Other Sheets and Models window

- Click **View** tab ► **Workspace** panel ► **Windows** drop-down, and select or clear the **Find Item in Other Sheets and Models** check box.

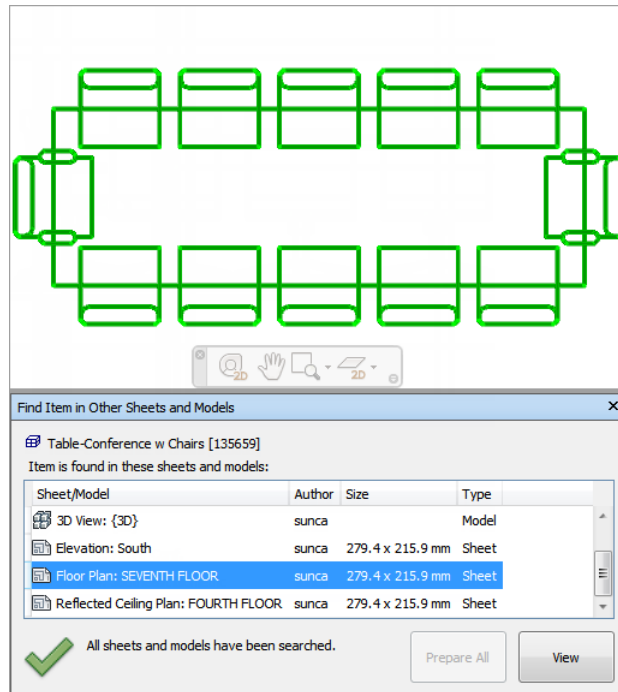
To find all sheets and models containing the selected object

- 1 Click  on the Status bar to open the **Project Browser** window.

- 2 Double-click the sheet or model with the object you want to locate in other sheets and models, and select it in the **Scene View** .



- 3 Right-click the object, and click **Find Item in Other Sheets and Models** on the shortcut menu.
- 4 If you see a warning status icon  in the **Find Item in Other Sheets and Models** window, click the **Prepare All** button. Once all sheets/models have been prepared, you will see a list of all sheets/models that contain the object.
- 5 Select the sheet/model in the list, and click **View** to open it. Autodesk Navisworks zooms you to the selected object.



Buttons

Prepare All Prepares all unprepared sheets and models in the currently open multi-sheet file.

View Opens the currently selected sheet or model in the **Scene View**.

Status Icons

 Not all sheets/models can be searched in the currently open multi-sheet file; at least one sheet or model has not yet been prepared.

 All sheets/models in the currently open multi-sheet file have been searched, and the results are up-to-date.

Create and Use Sets of Objects

In Autodesk Navisworks, you can create and use sets of like objects. This makes it easier to review and analyze your model.

Selection Sets

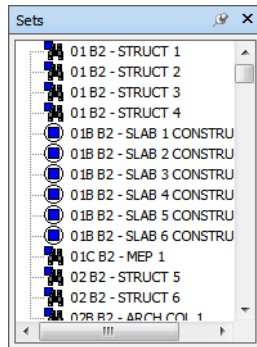
Selection sets are static groups of items, and are useful for saving a group of objects that you want to regularly perform some action on, such as hiding them, changing transparency and so on. They simply store a group of items for later retrieval. There is no intelligence behind this set - if the model changes at all, the same items are selected (assuming they are still available in the model) when recalling the selection set.

Search Sets

Search sets are dynamic groups of items, and work in a similar way to selection sets, except that they save search criteria instead of the results of a selection, so that you can re-run the search at a later date as and when the model changes. Search sets are much more powerful and can save you time, especially if your CAD files are continuing to be updated and revised. It is also possible to export search sets, and share them with other users.

Sets Window

The **Sets** window is a dockable window, which displays both selection sets and search sets available in your Autodesk Navisworks file.



The selection sets are identified by this icon: , and the search sets are identified by this icon: .

NOTE The list of the items on the **Sets** window is exactly the same as the list on the **Sets** tab of the **Selection Tree**.

You can customize the names of selection and search sets, and add comments. You can copy and paste names from the **Sets** window. To do this, right-click an item in the **Sets** window, and click **Copy Name** on the context menu. Alternatively, you can click an item in the **Sets** window, and press CTRL + C. The name is now copied to the clipboard.

You can also show the selection and search sets as [links](#) (page 421) in the **Scene View**. These links are created automatically by Autodesk Navisworks. Clicking on a link restores the geometry within the corresponding selection or search set to the active selection, and highlights it in the **Scene View** and on the **Selection Tree**.

You can use the **Sets** shortcut menu to create and manage selection and search sets in the Autodesk Navisworks file.

To toggle the Sets window

- Click View tab ► Workspace panel ► Windows drop-down, and select or clear the Sets check box.

 **Menu:** Classic user interface: View ► **Control Bars** ► Selection Sets
 **Command entry:** SHIFT + F2

To use the Sets window to select objects

- 1 Open the **Sets** window.
- 2 Click the desired set in the list. The geometry in this set is selected in the **Scene View** and in the **Selection Tree**.

NOTE If you clicked on a search set, the search is performed on the current model, looking for all geometry that meets the search criteria, including any additional geometry that could have been added to the file.

 **Menu:** Classic user interface: **Edit** ► **Select** ► **Sets** ► <Saved Set>

To change the sort order on the Sets window

- 1 Open the **Sets** window.

- 2 Right-click any item in the list, and click **Sort**. The contents of the tab is now ordered alphabetically.

The **Sets** shortcut menu has the following options:

- **New Folder.** Creates a folder above the selected item.
- **Add Current Selection.** Saves the current selection as a new selection set in the list. This set contains all currently selected geometry.
- **Add Current Search.** Saves the current search as a search set in the list. This set contains the current search criteria.
- **Make Visible.** If the geometry in the selected search or selection set is hidden, you can use this option to make it visible.
- **Add Copy.** Creates a copy of the search or selection set highlighted in the list. The copy has the same name as the original set, but with an “X” suffix, where “X” is the next available number.
- **Add Comment.** Opens the **Add Comment** dialog box for the selected item.
- **Edit Comment.** Opens the **Edit Comment** dialog box for the selected item.
- **Update.** Updates the selected search set with the current search criteria, or updates the selected selection set with the currently selected geometry.
- **Delete.** Deletes the selected search or selection set.
- **Rename.** Renames the selected search or selection set. By default, new selection sets are named **Selection SetX**, and search sets are named “Search SetX”, where “X” is the next available number added to the list.
- **Copy Name.** Copies the name of the search set to the clipboard.
- **Sort.** Orders the contents of the **Sets** window alphabetically.
- **Help.** Launches the online Help system and displays the topic for selection and search sets.

Create and Manage Selection and Search Sets

You can add, move and delete selection and search sets, and organize them into folders.

Search and selection sets can be updated. You can modify your current selection in the **Scene View**, or the current search criteria, and change the contents of your set to reflect this.

You can also export search sets and reuse them. For example, if models contain the same components, such as steel structure, ventilation ductwork, and so on, you can define generic search sets, export them as an XML file, and share with other users.

To save a selection set

- 1 Select all the items that you want to save in the **Scene View** or on the **Selection Tree**.
- 2 Click **Home** tab ► **Select & Search** panel ► **Save Selection** .
- 3 Type a name for your selection set in the **Sets** window, and press ENTER.

 **Menu:** Classic user interface: Edit ► Select ► Sets ► Add Current Selection

To save a search set

- 1 Open the **Find Items** window, and set up the desired search criteria.
- 2 Click the **Find All** button to run the search. All items that satisfy your criteria are now selected in the **Scene View** and the **Selection Tree**.
- 3 Open the **Sets** window, right-click, and click **Add Current Search**.
- 4 Type a name for your search set, and press ENTER

 **Menu:** Classic user interface: Edit ► Select ► Sets ► Add Current Search

To rename a search or selection set

- 1 Open the **Sets** window.
- 2 Right-click the desired search or selection set, and click **Rename**.
- 3 Type a new name for your set, and press ENTER.

 **Command entry:** F2

To delete a search or selection set

- 1 Open the **Sets** window.
- 2 Right-click the search or selection set you want to remove, and click **Delete**.

To organize selection and search sets into folders

- 1 Open the **Sets** window.
- 2 Right-click, and click **New Folder** on the shortcut menu. The folder is added to the list.

If, when you right-click, the item selected is a folder, then the new folder is created inside it, otherwise it is added above the selected item. You can have as many folders as you like.

- 3 Type a name for your folder, and press ENTER.

TIP Use names that can help you to identify your sets in future.

- 4 Click the set that you want to add to your new folder. Hold down the left mouse button and drag the mouse to the folder name. Release the mouse button to drop the set into the folder.

TIP You can also drag the folders themselves, together with their contents.

To update a selection set

- 1 Select the desired geometry in the **Scene View** or on the **Selection Tree**.
- 2 Open the **Sets** window.
- 3 Right-click the selection set you want to modify, and click **Update**.

To update a search set

- 1 Open the **Find Items** window, and run a new search.
- 2 Open the **Sets** window.
- 3 Right-click the search set you want to modify, and click **Update**.

To export search sets

- 1 Click **Output** tab ► **Export Data** panel ► **Search Sets** .
- 2 In the **Export** dialog box, browse to the desired folder.
- 3 Enter a name for your file, and click **Save**.

To import search sets

- 1 Click the application button  ► **Options**.
- 2 In the **Import** dialog box, browse to the folder containing the file with the search sets, and select it.
- 3 Click **Open**.

Compare Objects

You can look for differences between any two selected items in the scene. These items can be files, layers, instances, groups, or just geometry.

You can also use this feature to investigate the differences between two versions of the same model.

During the comparison, Autodesk Navisworks starts at the level of each item, and recursively travels down each path on the **Selection Tree**, comparing each item it comes across in terms of the criteria you requested.

When the comparison is finished, the results can be highlighted in the **Scene View**. The following color-coding is used by default:

- **White**. Matching items.
- **Red**. Items with differences.
- **Yellow**. The first item contains things not found in the second item.
- **Cyan**. The second item contains things not found in the first item.

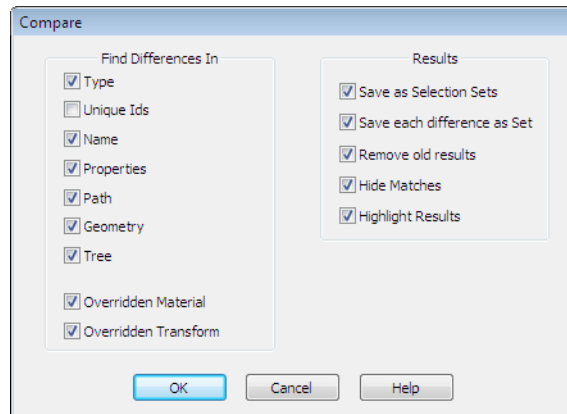
You can save the comparison results as selection sets with automatic comments describing the found differences in more detail.

To compare two versions of the model

- 1 Open the first file that you want to compare in Autodesk Navisworks.
- 2 Click **Home** tab ► **Project** panel ► **Append** drop-down ► **Append** , locate the second file, and click **Open**.
- 3 Hold down CTRL key, and select both files.
- 4 Click **Home** tab ► **Tools** panel ► **Compare** .
- 5 In the **Compare** dialog box, the **Find Differences In** area, select the check boxes for all required options.

NOTE The **Overridden Material** and **Overridden Transform** check boxes relate to changing the color and transparency in Autodesk Navisworks, and changing a file's origin, scale or rotation since loading into Autodesk Navisworks, respectively. These check boxes are clear by default. All the other criteria relate to properties of items from the original CAD model.

- 6 In the **Results** area, select the check boxes to control how the comparison results are displayed:
 - **Save as Selection Sets.** Saves the items that you are comparing as a selection set. You can then use this set for later comparisons between the same items.
 - **Save Each Difference As Set.** Saves the resulting differences found in the comparison between the two items as a selection set for later analysis. The selection set will also have a comment attached detailing the differences in more depth.
 - **Remove Old Results.** Removes any selection sets resulting from a previous comparison, in order to reduce confusion when looking at the results.
 - **Hide Matches.** Hides all items that turn out to be the same in the comparison, when the comparison finishes.
 - **Highlight Results.** Highlights each resulting difference with a color override, when the comparison finishes. You can reset the colors back by clicking **Home** tab ► **Project** panel ► **Reset All** drop-down ► **Appearances** .



- 7 Click **OK**.

To compare two items in the model

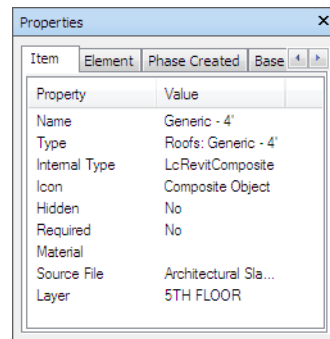
- 1 Select exactly two items in the scene.
- 2 Click **Home** tab ► **Tools** panel ► **Compare** .
- 3 Use the **Compare** dialog box to set up the comparison search.
- 4 Click OK.

Object Properties

Autodesk Navisworks supports the conversion of object properties for many file formats. For more details on supported/unsupported entities, see [Use File Readers](#) (page 170).

Properties Window

Once brought into Autodesk Navisworks, the object properties can be examined in the **Properties** window.



The **Properties** window is a dockable window, which has a dedicated tab for each property category associated with the currently selected object.

Internal file properties, such as transform and geometry properties, are not shown by default. The **Options Editor** enables you to switch this on.

You can use the **Properties** shortcut menu to create and manage custom object properties, and [links](#) (page 421).

You can also bring more object properties into Autodesk Navisworks from external databases, and show on the database-specific tabs in the **Properties** window.

To toggle the Properties window

- Click **View** tab ► **Workspace** panel ► **Windows** drop-down, and select or clear the **Properties** check box.

 **Menu:** Classic user interface: **View** ► **Control Bars** ► **Properties**

 **Command entry:** SHIFT + F7

To examine object properties

- 1 Select the object of interest in the **Selection Tree**, or in the **Scene View**.
- 2 Open the **Properties** window, and use the tabs to navigate between the available property categories.

NOTE If more than one object is selected, the **Properties** window only shows the number of selected items, and doesn't show any property information.

To add tabs with internal properties to the Properties window

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Developer** option.
- 3 On the **Developer** page, select the **Show Internal Properties** check box.
- 4 Click **OK**.

The Properties shortcut menu has the following options:

- **Copy Value** Copies the selected value.
- **Copy Row** Copies the selected row.
- **Copy All** Copies all on the tab.
- **Edit Links**. Opens the **Edit Links** dialog box.
- **Add New User Data Tab**. Adds a custom property tab to the **Properties** window.

- **Delete New User Data Tab.** Deletes the currently selected custom property tab from the **Properties** window.
- **Insert New Property.** Adds a custom property.
- **Delete Property.** Deletes the currently selected custom property.
- **Edit Property Value.** Enables you to edit the value for all property types except Boolean.
- **Set Value.** Enables you to edit the value for Boolean property type.
- **Rename Property.** Renames the currently selected custom property.
- **Rename Tab.** Renames the currently selected custom property tab on the **Properties** window.

Custom Properties

Property information brought into Autodesk Navisworks from a CAD application cannot be edited, with the exception of color, transparency and links. You can, however, add your own custom information to any item in the model scene.

To add a custom property tab

- 1 Open the **Properties** window.
- 2 Select the object of interest in the **Scene View** or on the **Selection Tree**.
- 3 Right-click the **Properties** window, and click **Add New User Data Tab**. The new property category is added for the currently selected object. By default, the tab is called **User Data**.

To rename a custom property tab

- 1 Open the **Properties** window.
- 2 Click the custom tab you want to rename.
- 3 Right-click the properties area, and click **Rename Tab**.
- 4 Enter a new name for the tab in the box provided.
- 5 Click **OK**.

To delete a custom property tab

- 1 Open the **Properties** window.

- 2 Click the custom tab you want to delete.
- 3 Right-click the properties area, and click **Delete User Data Tab**.

To add a custom property

- 1 Open the **Properties** window.
- 2 Click the custom tab where you want to add a property.
- 3 Right-click the properties area, click **Insert New Property**, and click the property type you want to add. Choose:
 - **String** - if the property value is a string
 - **Boolean** - if the property value is Yes or No
 - **Float** - if the property value is a decimal value, for example: 1.234
 - **Integer** - if the property value is a positive whole number (1, 2, 3,...), a negative whole number (-1, -2, -3,...), or zero (0)
- 4 Enter the name for the property in the **Property** box, and press ENTER.
- 5 Double-click the **Value** box, enter the value associated with your property into the box provided, and click **OK**.

NOTE For Boolean properties, double-clicking the **Value** box toggles between Yes and No values.

To edit a custom property

- 1 Open the **Properties** window.
- 2 Click the custom tab where the property you want to edit is located.
- 3 To rename the property, right-click it, click **Rename Property**, and enter the new name in the box provided. Click **OK**.
- 4 To edit the property value, right-click it, click **Edit Property Value**, and enter the new value in the box provided. Click **OK**.

NOTE For Boolean properties, click **Set Value** on the shortcut menu, and choose Yes or No.

To delete a custom property

- 1 Open the **Properties** window.
- 2 Click the custom tab where the property you want to delete is located.
- 3 Right-click the property, and click **Delete Property**.

External Database Links

Databases are commonly used to store large amount of data, such as equipment specifications, catalogue data, and maintenance manuals.

You can connect to external databases directly from your Autodesk Navisworks files, and create links between objects in the scene and fields in the database tables to bring through extra properties.

Any database with a suitable ODBC driver is supported, but the properties for the objects in the model must include unique identifiers to the data in the database. For example, for AutoCAD-based files, entity handles can be used.

You can have as many database links as you want, but they all should have unique names. To use a database link, you need to activate it first.

Database links can be saved inside Autodesk Navisworks files (NWF and NWD). You can also save database links globally, making them persistent across all Autodesk Navisworks sessions. The global connection information is saved on the local machine. If the associated database is available on loading the NWF/NWD file, the link automatically establishes itself when an object is selected. On selecting an object, if the database is available, and there is data associated with the object, Autodesk Navisworks adds an appropriate database tab to the **Properties** window, and displays the appropriate data.

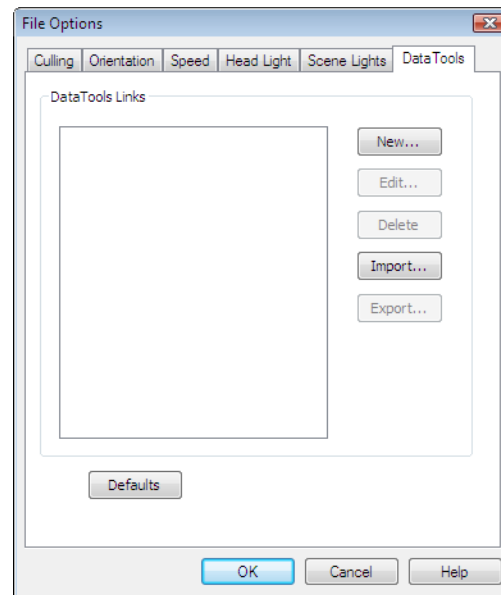
NOTE To be able to display the data, you need to configure the connection details. You may need a database administrator to do this, as a knowledge of the database being connected to is required. You can use Autodesk Navisworks tags in connection strings and SQL statements.

Data linked from a database connection can be extracted and embedded as static data within the published NWD file. It also can also be included in object searches, and can be used with the **Clash Detective** tool as part of the clash criteria.

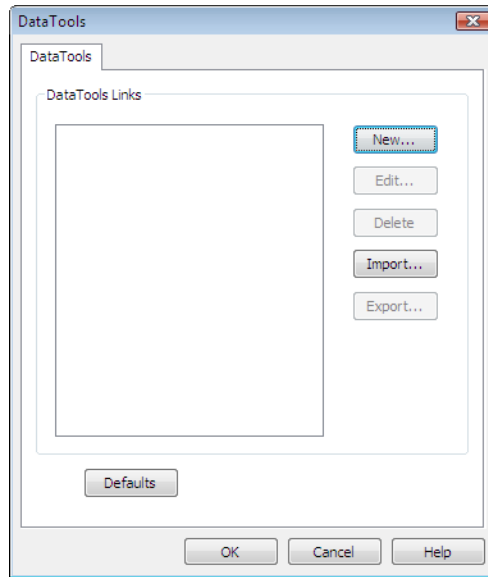
You can export database links, and share them with other users.

To add a database link

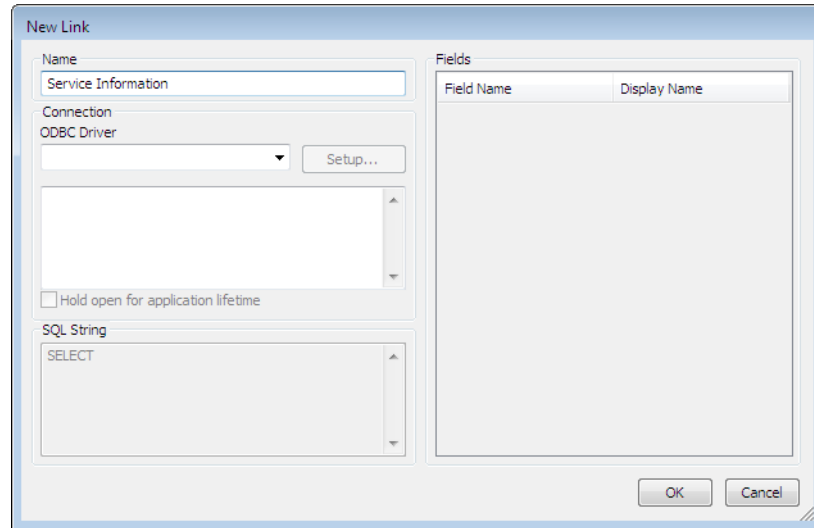
- 1 If you want to add a database link to the Autodesk Navisworks file:
 - 1 Click **Home** tab ► **Project** panel ► **File Options** .
 - 2 In the **File Options** dialog box, **DataTools** tab, click the **New** button.



- 2 If you want to add a database link globally:
 - a Click **Home** tab ► **Tools** panel ► **DataTools** .
 - b In the **DataTools** dialog box, click the **New** button.



- 3 In the **New Link** dialog box, enter a **Name** for the new link, for example, 'Service Information'. This is the name of the tab that will appear on the **Properties** window.

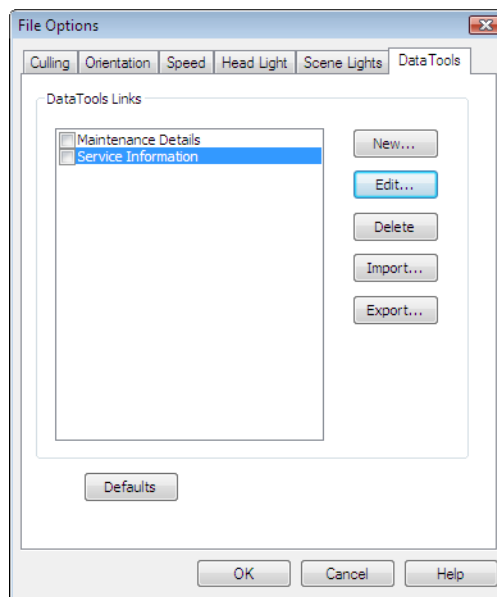


You are now ready to configure the database link.

- 4 Click **OK** to save the link and return to the previous dialog box.

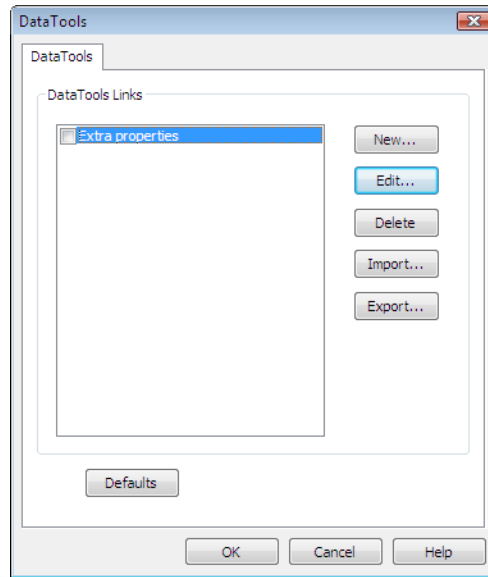
To configure a database link

- 1 To configure a file-based database link:
 - 1 Click **Home** tab ► **Project** panel ► **File Options** .
 - 2 In the **File Options** dialog box, **DataTools** tab, click the link you want to configure in the **DataTools Links** area, and click the **Edit** button.



NOTE If you haven't added any links yet, click the **New** button, and follow the procedure for adding database links first.

- 2 To configure a global database link:
 - A Click **Home** tab ► **Tools** panel ► **DataTools** .
 - B In the **DataTools** dialog box, click the link you want to configure in the **DataTools Links** area, and click the **Edit** button.



NOTE If you haven't added any links yet, click the **New** button, and follow the procedure for adding database links first.

3 Configure the ODBC Driver:

- a In the **Edit Link** dialog box, select the appropriate ODBC Driver to define the type of the database to link to, for example Microsoft Access Driver (*.mdb).
- b Click the **Setup** button. The driver wizard opens, and guides you through the setup options. If you have difficulties setting up your connection details, contact your database administrator. When you finish, the box underneath will show the connection string. This string can be modified, if necessary.

For example, if you want to select a database with a certain name (say, test.mdb) that is always next to the model file (as in the resultant NWD/NWF not the original), type in:

```
DBQ=%pushpath(%poppath(%currentpath),"test.mdb");
DRIVER={Microsoft Access Driver (*.mdb)};
```

If you want to select a database with the same name as the original model file (say, AutoPlant), type in:

```
DBQ=%join(%removeext(%removepath(%sourcepath)),".mdb");
DRIVER={Microsoft Access Driver (*.mdb)};
```

- 4 Select the **Hold Open for Application Lifetime** check box, if you want to keep the database link open until you exit Autodesk Navisworks.

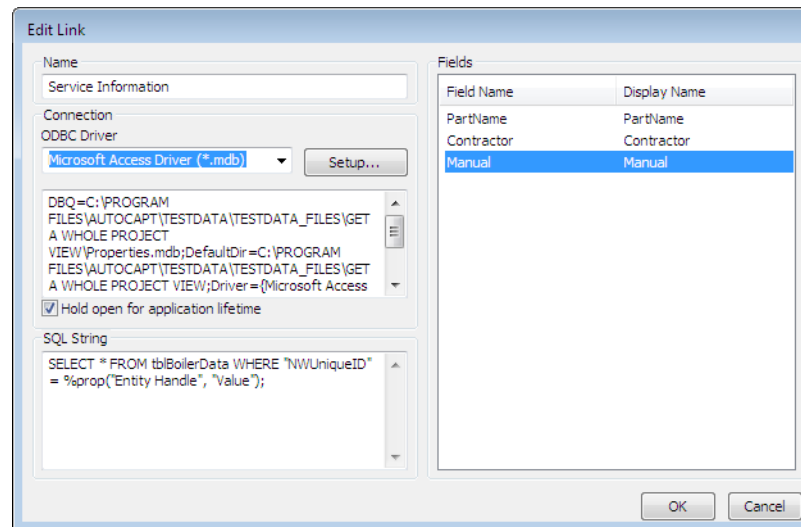
- 5 Specify which table in the database to query:

In the **SQL String** field, click after **SELECT**, and enter the selection statement, for example: `SELECT * FROM tblBoilerData WHERE "NWUniqueID" = %prop("Entity Handle", "Value");`

This statement instructs Autodesk Navisworks to select all columns from the `tblBoilerData` table, whilst requiring that the column called `NWUniqueID` matches a category/property pair called `Entity Handle/Value`.

- 6 Select which columns you want to display as link categories on the **Properties** window:
 - a Double-click the **Fields Name** field, and type the exact name of the database column, for example: "Part-time".
 - b Press ENTER.

The **Display Name** is automatically completed for you, but you can click it, and enter a different name, if you want. The text entered here is the category name shown on the link tab of the Properties window.
 - c Repeat the above process to list all required categories in the **Fields** area of the dialog box.



- 7 Click OK.

IMPORTANT The model is not linked to the external data source until you have activated the database link.

To delete a database link

- 1 To delete a file-based database link:
 - a Click **Home** tab ► **Project** panel ► **File Options** .
 - b In the **File Options** dialog box, **DataTools** tab, click the link you want to remove in the **DataTools Links** area, and click the **Delete** button.
- 2 To delete a global database link:
 - a Click **Home** tab ► **Tools** panel ► **DataTools** .
- 3 In the **DataTools** dialog box, click the link you want to remove in the **DataTools Links** area, and click the **Delete** button.
- 4 Click **OK**.

To activate a database link

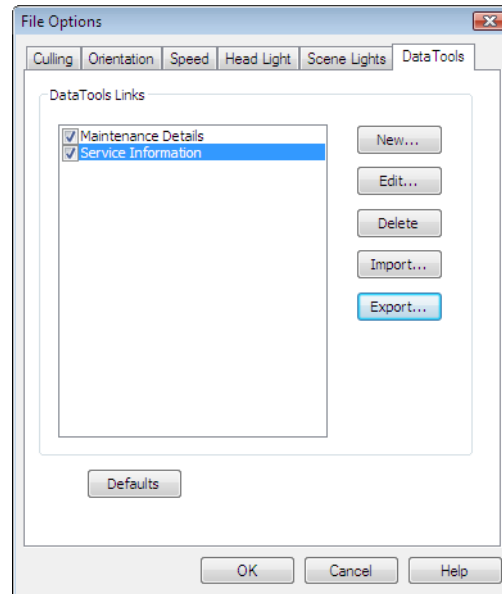
- 1 To activate a file-based database link:
 - a Click **Home** tab ► **Project** panel ► **File Options** .
 - b In the **File Options** dialog box, **DataTools** tab, select the check boxes for all links you want to activate in the **DataTools Links** area.
- 2 To activate a global database link:
 - a Click **Home** tab ► **Tools** panel ► **DataTools** .
 - b In the **DataTools** dialog box, select the check boxes for all links you want to activate in the **DataTools Links** area.
- 3 Click **OK**.

IMPORTANT You cannot activate links with insufficient or invalid configuration information.

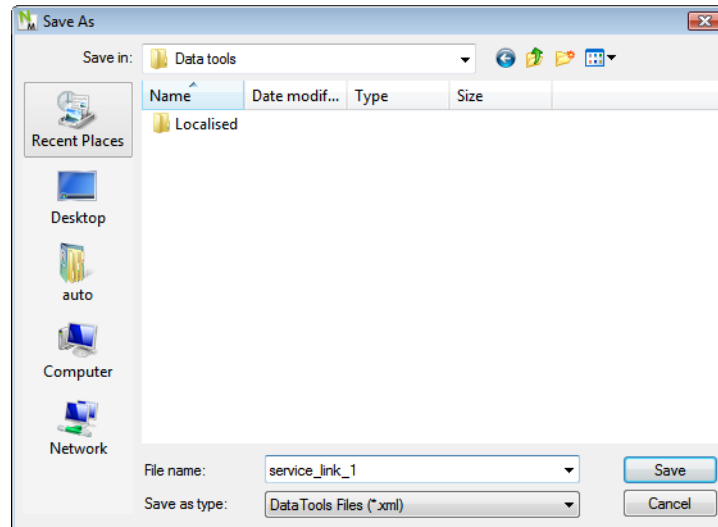
To export a database link

- 1 Click **Home** tab ► **Project** panel ► **File Options** .

- 2 In the **File Options** dialog box, **DataTools** tab, click the link you want to export in the **DataTools Links** area, and click the **Export** button.



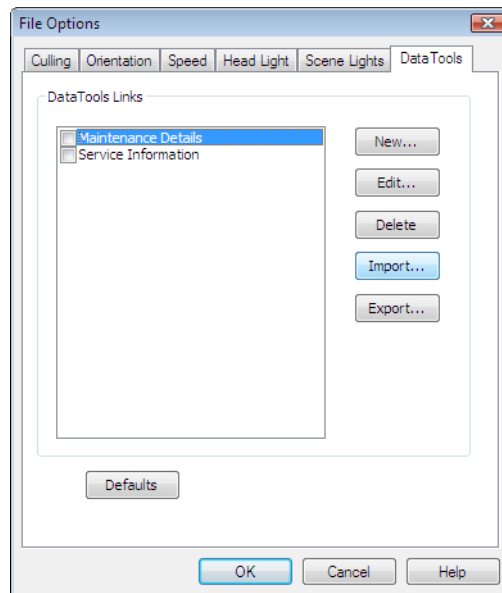
- 3 In the **Save As** dialog box, browse to the desired folder, and enter a name for the datatools file. You can also select the name of an existing datatools file to overwrite it with your modified configuration.



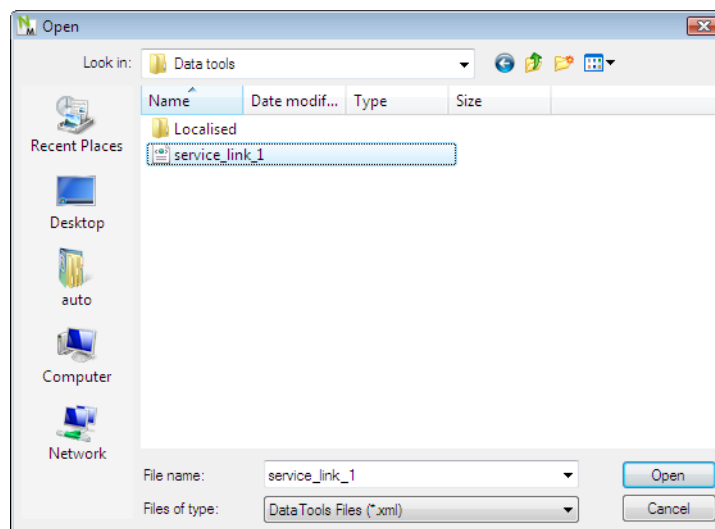
- 4 Click **Save**.
- 5 Click **OK** to close the **File Options** dialog box.

To import a database link

- 1 Click **Home** tab ► **Project** panel ► **File Options** .
- 2 In the **File Options** dialog box, **DataTools** tab, click the **Import** button.



- 3 In the **Open** dialog box, browse to the folder containing the required datatools file, select it, and click **Open**.



- 4 Click **OK** to close the **File Options** dialog box.

You can use the following Autodesk Navisworks tags with SQL queries:

Property Tags

%prop("category","property") Property of the currently selected object. Category is the name of the tab in the property windows (for example, Item or Entity Handle) and property is the name of the property in that tab (for example, Value or Layer).

%intprop("category","property") Property of the currently selected object. This is the same as the previous tag except instead of using the publicly visible category and property name use the internal Autodesk Navisworks names. The benefit of using internal names is that they are not language dependent. This is advanced tag suitable for users familiar with the Autodesk Navisworks API.

File and Path Tags

%sourcepath This tag represents the full path and filename that the currently selected object comes from. Even when a collection of model files have been combined into a single NWD file this tag still remembers the path and filename of the original model file.

%currentpath This tag represents the full path and filename of the currently loaded model. If you currently have an NWF or NWD loaded that contains many other models it will just return the path and filename of the top level NWF/NWD.

File and Path Manipulation Tags

%removeext("text") If the provided text includes a filename with an extension, this tag removes the extension.

%removepath("text") If the provided text includes a path and filename, this tag remove the paths and just returns the filename.

%poppath("text") If the provided text includes a path, this tag removes the top level. If it the text also includes a filename, it counts as the top level and is removed. For example, `%poppath("c:\temp")` becomes `c:\` and `%poppath(c:\temp\readme.txt)` becomes `c:\temp`.

%pushpath("text1","text2") If `text1` is a path and `text2` is a file or folder name, then `text2` is added onto the path in `text1`. For example, `%pushpath("c:\test","model.nwd")` becomes `c:\test\model.nwd`.

String Manipulation Tags

`%join("text","text")` This tag simply joins the two pieces of text together. For example, `%join("c:\", "model.nwd")` returns `c:\model.nwd`.

Query Examples

The examples below illustrate how tags could be used with SQL queries.

- Selecting all columns from table `Test` whilst requiring that the column called `Entity Handle` matches a category/property pair called `Entity Handle/Value` and the column called `File Name` matches the original filename of the drawing:

```
SELECT * FROM Test WHERE "Entity Handle" = %prop("Entity  
Handle", "Value") AND "File Name" =  
$removeext(%removepath(%sourcepath));
```

Here the path and the extension of the file name are being stripped, so a file like `c:\model\3rdFloorDucts.dwg` would come out as `3rdFloorDucts`.

- Selecting two columns from table `Test` whilst requiring that the column called `Entity Handle` matches a category/property pair called `Entity Handle/Value`:

```
SELECT Name,Part FROM Test WHERE "Entity Handle" = %prop("Entity  
Handle", "Value");
```

- Selecting all columns from table `Test` whilst requiring that the column called `Value` is within a certain range given by two category/property pairs:

```
SELECT * FROM Test WHERE Value BETWEEN %prop("Pressure", "Minimum")  
AND %prop("Pressure", "Maximum");
```

NOTE Tags must not contain white space between the brackets (unless enclosed by quote marks), so `%prop("EntityHandle", "Value")` works, but `%prop("EntityHandle", "Value")` does not.

Manipulate Object Attributes

In Navisworks Manage, you can manipulate objects' transforms (translation, rotation, and scale), and also change appearance (color and transparency) of objects. All object manipulation is carried out in the **Scene View**.

Any changes that you make to object attributes are considered to be global, (as if they'd been changed in the original CAD model), and can be saved with

Autodesk Navisworks files. You have an option of resetting object attributes back to the state they were in when imported from the original CAD files.

Manipulating Object Attributes During Object Animation

You can temporarily modify position, rotation, size, and appearance of geometry objects for animation purposes. These changes are not global, and can only be saved (or captured) as animation keyframes.

Transform Objects

To transform objects, you can use three visual manipulation tools, or gizmos, available from the Item Tools tab ► Transform panel. You can also transform objects numerically.

To get a clearer view of objects as you manipulate them, you can use the **Options Editor** to adjust the way in which the current selection is highlighted. For more information, see [Set Highlighting Method](#) (page 346).

Using Measure Tools to Transform Objects

You can use the [Measure Tools](#) (page 391) functionality to move and rotate the currently selected objects.

To move an object with gizmo

- 1 Select the object you want to move in the **Scene View**.
 - 2 Click Item Tools tab ► Transform panel ► Move .
 - 3 Use the move gizmo to adjust the position of the currently selected object:
 - To move all currently selected objects, place the mouse over the arrow at the end of the desired axis. When the cursor changes to , drag the arrow on the screen to increase/decrease the translation along that axis.
 - To move the objects along several axes at the same time, drag the square frame between the desired axes.
- Dragging the ball in the middle of the move gizmo enables you to snap this center point to other geometry in the model.

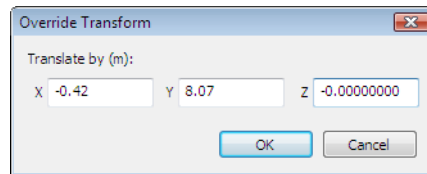
- To move the gizmo itself rather than the selected objects, hold down the CTRL key while dragging the ball in the middle of the gizmo. The move gizmo will snap to other objects as you do this.

To move an object numerically

- 1 Select the object you want to move in the **Scene View**.
- 2 Click the Item Tools tab, and slide out the Transform panel.
- 3 Type in numerical values into the manual entry boxes to move the object by the amount entered:
 - Position X, Y, Z represent translation distance in the current model unit.
 - Transform Center X, Y, Z represent the translation center point.

To move an object by overriding its transform (classic user interface only)

- 1 Select the object you want to move in the **Scene View**.
- 2 Click Edit ► Override Item ► Override Transform.
- 3 In the **Override Transform** dialog box, enter the XYZ values of the transform to be applied to the currently selected object. For example, a transform of (0, 0, 1) repositions an object by one unit in the Z direction.



- 4 Click OK.

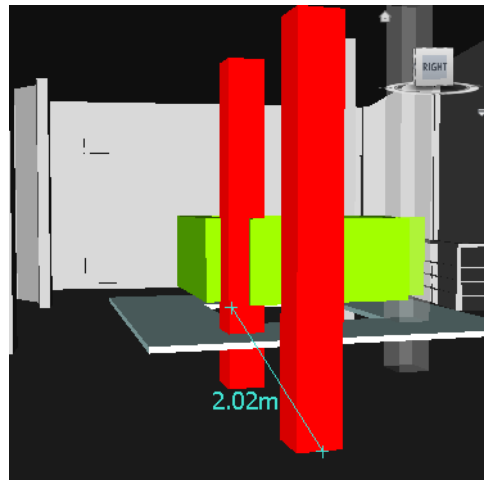
To move an object with a measure tool

- 1 Select the object you want to move.
- 2 Click Review tab ► Measure panel ► Measure drop-down ► Point Line  .
- 3 Click the selected object to create the first point. This is the start point from which the repositioning will be calculated.

- 4 Click the second point in the scene. This is the point where the object will be moved to. There is now a line connecting the start point and the end point in the **Scene View**.
- 5 If you want to be able to move the object several times, create more points in the scene.

NOTE You can only select a point on another object in the scene. Selecting a point in 'space' is not a valid option. To reposition an object into 'space', you can either use a translation gizmo or, if you know the distance by which the object is to be moved, by overriding its transform.

- 6 Slide out the **Measure** panel, and click **Transform Selected Items**  to move the object to the second point. If you have multiple points in the scene, each time you click **Transform Objects** the selected object is moved to the next point.



To rotate an object with gizmo

- 1 Select the object you want to rotate in the **Scene View**.
- 2 Click Item Tools tab ➤ Transform panel ➤ Rotate .
- 3 Use the gizmo to rotate the currently selected object:
 - Before you can rotate the currently selected objects, you need to position the origin (center point) of the rotation. To do this, place the mouse over the arrow at the end of the desired axis. When the cursor changes to , drag the arrow on the screen to

increase/decrease the translation along that axis. This will move the gizmo itself.

Dragging the ball in the middle of the rotate gizmo enables you to move it around, and snap it to points on other geometry objects.

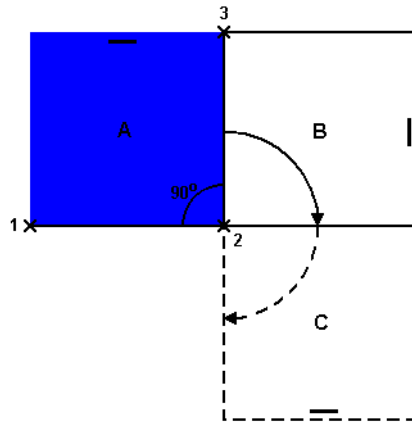
- Once the rotate gizmo is positioned correctly, place the mouse over one of the curves in the middle, and drag it on the screen to rotate the selected objects. The curves are color-coded, and match the color of the axis used to rotate the object around. So, for example, dragging the blue curve between the X and Y axes, rotates the objects around the blue Z axis.
- To rotate the orientation of the gizmo to an arbitrary position, hold down the CTRL key while dragging one of the three curves in the middle.
- To snap the gizmo to other objects, hold the CTRL key while dragging the ball in the middle of the gizmo.

To rotate an object numerically

- 1 Select the object you want to rotate in the **Scene View**.
- 2 Click the Item Tools tab, and slide out the Transform panel.
- 3 Type in numerical values into the manual entry boxes to move the object by the amount entered:
 - Rotation X, Y, Z represent degrees of rotation in the current model unit.
 - Transform Center X, Y, Z represent the rotation center point.

To rotate an object with a measure tool

- 1 Select the object you want to move.
- 2 Click Review tab ► Measure panel ► Measure drop-down ► Measure Angle .
- 3 To rotate the object, click three points on the selected object to determine how the object is to be rotated (refer to the diagram below).



This diagram illustrates how the selected object (position A) is rotated. Lines from the first to the second point (1 to 2) and from the second to third point (2 to 3) define the angle by which the object is rotated (for example, 90 degrees). The second point (2) is used as the rotation center point.

- 4 Slide out the **Measure** panel, and click **Transform Selected Items** . This rotates the selected object from position A into position B (as shown in the previous diagram). Clicking this button again, rotates the object from position B into position C.

To resize an object with the gizmo

- 1 Select the object you want to resize in the **Scene View**.
- 2 Click Item Tools tab ➤ Transform panel ➤ Rotate .
- 3 Use the scale gizmo to resize the currently selected object:
 - To resize the objects across a single axis only, use colored arrows at the end of the axes. To resize the objects across two axes at the same time, use colored triangles in the middle of the axes. Finally, to resize the objects across all three axes at the same time, use the ball in the center of the gizmo.
 - You can modify the center of scaling. To do this, place the mouse over the ball in the middle of the gizmo, and hold down the CTRL key while dragging the ball on the screen.

To resize an object numerically

- 1 Select the object you want to resize in the **Scene View**.
- 2 Click the Item Tools tab, and slide out the Transform panel.
- 3 Type in numerical values into the manual entry boxes to move the object by the amount entered:
 - Scale X, Y, Z represent a scaling factor (1 being the current size, 0.5 half, 2 being double, and so on).
 - Transform Center X, Y, Z represent the scaling center point.

Change Object Appearance

You can apply custom colors and transparencies to geometry in the scene.

For example, when you are dealing with models with unsupported materials and textures, all geometry is displayed in its wireframe color. You can change (or override) the appearance of objects in the scene for more realistic presentation.

You can also choose to use **Presenter** tool to apply texture materials to objects in the scene to obtain even better results.

NOTE Any materials applied with the **Presenter** tool supersede any color and transparency changes.

To change color

- 1 Select the object you want to modify in the **Scene View**.
- 2 Click Item Tools tab ➤ Appearance panel ➤ Color drop-down, and choose the desired color.

 **Menu:** Classic user interface: Edit ➤ Override Item ➤ Override Color

To change transparency

- 1 Select the object you want to modify in the **Scene View**.
- 2 Click Item Tools tab ➤ Appearance panel.
- 3 Move the Transparency slider to adjust how transparent or opaque the selected object is.

 **Menu:** Classic user interface: Edit ► Override Item ► Override Transparency

Snapping

Snapping gives you control when measuring, moving, rotating, and scaling objects in Autodesk Navisworks.

Points and snap points are automatically snapped to. You can set the cursor to snap to the nearest vertex, edge, or line when you pick geometry. You can also adjust the snapping angle and the snapping tolerance that are used when you rotate geometry.

Different cursors feed back what is being snapped to:

Cursor	Description
	No snap, but a point on a surface is found.
	A vertex, point, snap point or line end is found to snap to.
	An edge is found to snap to.

Geometry in Autodesk Navisworks is tessellated with triangles, and therefore, the cursor snaps to edges that may appear to be in the middle of a face.

Consider viewing the model in hidden line mode (Viewpoint tab ► Render

Style panel ► Mode drop-down ► Hidden Line ) to clarify which vertex or edge the cursor is snapping to.

To toggle snapping

- Click the Item Tools tab, slide out the Transform panel, and click Snap Item .

To customize snap settings

- 1 Click the application button  ► **Options**.

- 2 In the **Options Editor**, expand the **Interface** node, and click the **Snapping** option.
- 3 On the **Snapping** page, **Picking** area, select the check boxes for all required snaps, and enter the snapping **Tolerance**. The smaller the value, the closer the cursor must be to a feature in the model before it snaps to it.
- 4 In the **Rotation** area, enter the multiplier for the snapping angle into the **Angles** box, and the snapping tolerance into the **Angle Sensitivity** box. The smaller the value, the closer to the snapping angle the cursor must be for snap to take effect.
- 5 Click OK.

Reset to Original Values

In Autodesk Navisworks, you can reset object attributes back to the values in the original CAD files.

To restore original appearance of an object or a group of objects in your scene

Note: You cannot restore colors separately from transparencies.

- 1 Select the required objects in the **Scene View**.
- 2 Click Item Tools tab ➤ Appearance panel ➤ Reset Appearance .

To reset appearance of all objects in your scene

Note: You cannot restore colors separately from transparencies.

- Click **Home** tab ➤ Project panel ➤ Reset All drop-down ➤ Appearances .

To reset transform for an object or a group of objects in your scene

- 1 Select the required objects in the **Scene View**.
- 2 Click Item Tools tab ➤ Transform panel ➤ Reset Transform .

To reset transform for all objects in your scene

■ Click **Home** tab ➤ Project panel ➤ Reset All drop-down ➤ Transforms



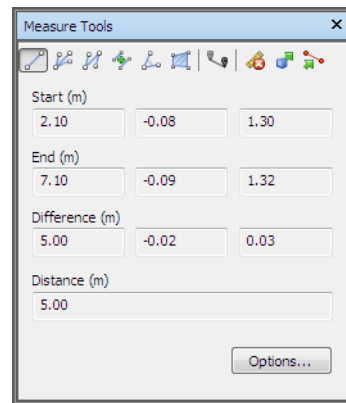
Measure Tools

Measure tools enable you to measure between points on items in the model. All measurements are made in [display units](#) (page 157).

Using measure tools is mutually exclusive to using navigation tools (see [Product-Specific Navigation Tools](#) (page 241)), so that when you are measuring you cannot navigate and vice versa.

Measure Tools Window

The **Measure Tools** window is a dockable window, which contains a number of buttons at the top enabling you to select the type of measurement you want to do.



For all measurements, the X, Y, and Z coordinates of the **Start** point and **End** point are displayed in the text boxes underneath the buttons, together with the **Difference** and the absolute **Distance**. If you use accumulative measure, such as Point Line or Accumulate, **Distance** shows the accumulated distance for all points registered in the measurement.

NOTE Z coordinate values are not available for 2D sheets.

To toggle the Measure Tools window

- Click Review tab ➤ Measure panel ➤ Measure Options tool launcher 

 **Menu:** Classic user interface: Tools ➤ Measure

Button	Description
	Measures the distance between two points.
	Measures the distance between a base point and various other points.
	Measures a total distance between multiple points along a route.
	Calculates the sum total of several point-to-point measurements.
	Calculates an angle between two lines.
	Calculates an area on a plane.
	Measures the shortest distance between two selected objects.
	Clears all measuring lines in the Scene View .
	Enables you to move or rotate an object.
	Converts the endpoint markers, the lines, and any displayed measurement values into redlines.

Measuring

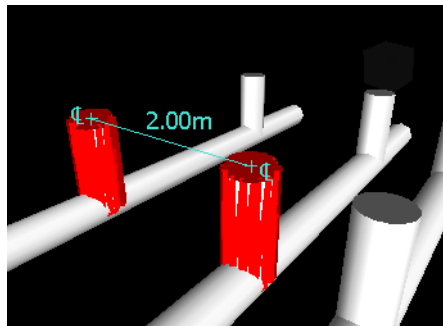
You can use measuring tools to make linear, angular, and area measurements, and to automatically measure the shortest distance between two selected objects.

NOTE When you measure, you must click on a point on an item to register a point - clicking on the background will not register anything. You can reset a measure command at any time by right-clicking instead of left-clicking in the **Scene View**. This starts the measure command again with no points registered, just as if you had chosen a new measurement type.

Endpoints of standard measuring lines are represented as small cross symbols in the **Scene View**, and all lines being measured by a simple line between registered points.



Endpoints of measuring lines that are snapped to center lines are represented as cross symbols with additional CL markers.



You can change the color and thickness of measuring lines, and toggle the display of dimension label in the **Scene View**.

Dimension Labels

For distance-based measurements, the dimension label is drawn for each line segment. For accumulative measurements, the dimension label shows the totals, and is drawn for the last line segment. The text is positioned relative to the center point of the line.

For angular measurements, an arc indicator is shown inside the angle, with the centre of the text positioned on the invisible line bisecting the angle. If an angle is too acute, the label is drawn outside the angle. This label is fixed, and does not resize when you zoom in or out, unless the measuring lines become too short on screen to accommodate the arc, in which case it will be adjusted.

The **Options Editor** enables you to toggle the dimension labels on and off.

For area measurements, the dimension label is positioned at the centre of the area being measured.

Converting Measurements to Redlines

You can convert measurements to redlines. The measurement itself is cleared when it is converted to a redline, and the redline takes the color and line thickness that is currently set for redlining.

NOTE When you convert measurements to redlines, the lines and text are stored in the current viewpoint.

To change the thickness and color of measuring lines

- 1 Open the **Measure Tools** window, and click **Options**.
- 2 In the **Options Editor**, the **Measure** page under the **Interface** node, enter the desired number into the **Line Thickness** box.
- 3 Select the required color from the **Color** palette. By default, measuring lines are white.
- 4 Click OK.

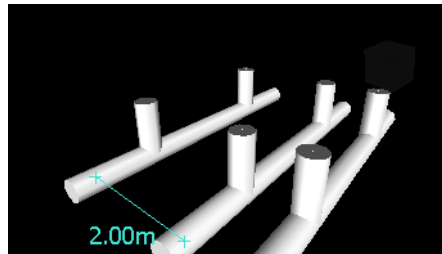
To toggle the dimension labels

- 1 Open the **Measure Tools** window, and click **Options**.
- 2 In the **Options Editor**, the **Measure** page under the **Interface** node, select the **Show Measurement Values in Scene View** check box.
- 3 Click OK.

To measure a distance between two points

- 1 Click Review tab ➤ Measure panel ➤ Measure drop-down ➤ Point to Point .
- 2 Click the start and end point of the distance to be measured in the **Scene View**.

The optional dimension label displays the measured distance.



 **Menu:** Classic user interface: Review ➤ Measure ➤ Point to Point

To keep the same start point as you measure a distance between two points

- 1 Click Review tab ➤ Measure panel ➤ Measure drop-down ➤ Point to Multiple Points .
- 2 Click the start point, and the first end point to be measured. A measuring line is displayed between two points.
- 3 Click to register the next end point to be measured.
- 4 Repeat this to measure additional end points if required. The optional dimension label always displays the last measured distance. Your start point stays the same throughout.

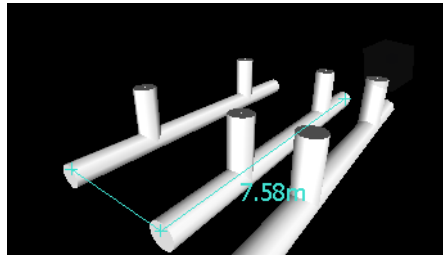
TIP If you want to change the start point, right-click in the **Scene View**, and select a new start point.

 **Menu:** Classic user interface: Review ➤ Measure ➤ Point to Multiple Points

To measure a total distance along a route

- 1 Click Review tab ➤ Measure panel ➤ Measure drop-down ➤ Point Line .

- 2 Click the start and the second point to be measured.
- 3 Click the next point along the route.
- 4 Repeat this to measure the entire route. The optional dimension label displays the total distance along the selected route.

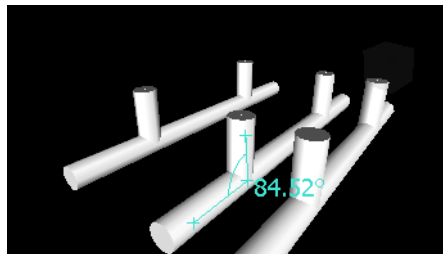


TIP If you want to change the start point, right-click in the **Scene View**, and select a new start point.

 **Menu:** Classic user interface: Review ► Measure ► Point Line

To calculate an angle between two lines

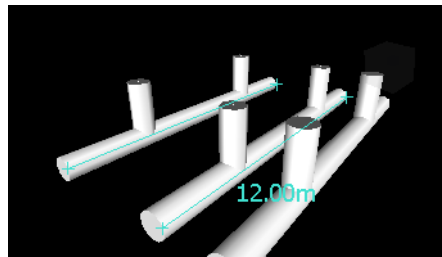
- 1 Click Review tab ► Measure panel ► Measure drop-down ► Angle .
- 2 Click a point on the first line.
- 3 Click the first line at the point where it intersects the second line.
- 4 Click a point on the second line. The optional dimension label displays the calculated angle between the two lines.



 **Menu:** Classic user interface: Review ► Measure ► Measure Angle

To calculate the sum total of several point-to-point measurements

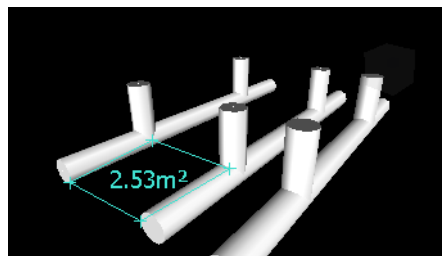
- 1 Click **Review** tab ► **Measure** panel ► **Measure** drop-down
► **Accumulate** .
- 2 Click the start and end points of the first distance to be measured.
- 3 Click the start and end points of the next distance to be measured.
- 4 Repeat to measure more distances if necessary. The optional dimension label displays the sum of all point-to-point measurements.



 **Menu:** Classic user interface: **Review** ► **Measure** ► **Accumulate**

To calculate an area on a plane

- 1 Click **Review** tab ► **Measure** panel ► **Measure** drop-down ► **Area**
.
- 2 Click to register a series of points to describe the perimeter of the area you want to calculate. The optional dimension label displays the area of the perimeter described since the first point, as projected onto the plane of the viewpoint.



NOTE For your calculation to be accurate, all added points must be on the same plane.

 **Menu:** Classic user interface: **Review** ➤ **Measure** ➤ **Measure Area**

To measure the shortest distance between two objects

- 1 Press and hold the CTRL key, and select two objects in the **Scene View** with the Select tool .

- 2 Click Review tab ➤ Measure panel ➤ Measure Shortest Distance .

The optional dimension label displays the shortest distance between the selected objects.

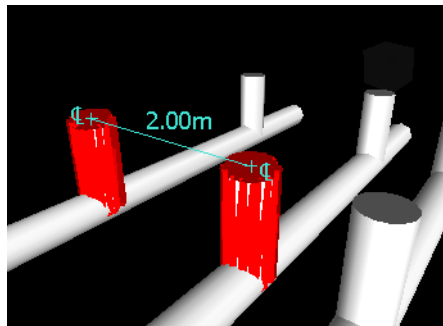
 **Menu:** Classic user interface: Review ➤ Measure ➤ Measure Shortest Distance

To measure the shortest distance between two parametric objects

- 1 Press and hold the CTRL key and select two parametric objects in the **Scene View** with the Select tool .
- 2 Open the **Measure Tools** window, and click **Options**.
- 3 In the **Options Editor**, the **Measure** page under the **Interface** node, select the **Use Center Lines** check box, and click OK.

- 4 Click Review tab ➤ Measure panel ➤ Measure Shortest Distance .

The **Distance** box, and the optional dimension label display the shortest distance between the center lines of the selected parametric objects.



 **Menu:** Classic user interface: Review ➤ Measure ➤ Measure Shortest Distance

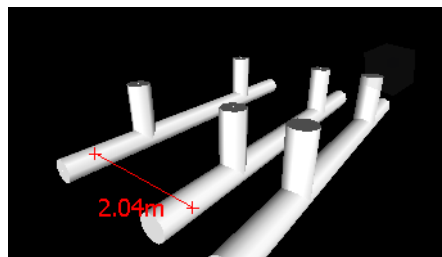
To clear measuring lines

- Click Review tab ► Measure panel ► Measure drop-down ► Clear 

 **Menu:** Classic user interface: Review ► Measure ► Clear

To convert measurements into redlines

- 1 Click Review tab ► Measure panel, and take the desired measurements, for example, distance between two points.
- 2 Click Review tab ► Measure panel ► Convert to Redline  .
The end markers, lines, and dimension label (if any) of your current measurement are now converted to a redline, and stored in the current viewpoint.



Comments, Redlines, and Tags

Use Comments, Redlines, and Tags

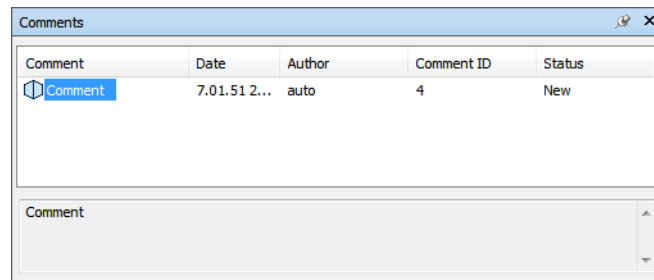
You can add comments to viewpoints, viewpoint animations, selection and search sets, clash results, and **TimeLiner** tasks.

Review tools (redlines and tags) enable you to add annotations to viewpoints, and clash detection results.

Using review tools is mutually exclusive to using navigation tools (see [Product-Specific Navigation Tools](#) (page 241)), so that when you add redlines or tags you cannot navigate and vice versa.

Comments Window

The **Comments** window is a dockable window that enables you to view and manage comments.



The **Comments** window shows the name, time and date, author, ID, status, and subject (or first line) of each comment. There are different icons helping you to identify the source of each comment at a glance.

The Comment shortcut menu has the following options:

- **Add Comment.** Opens the Add Comment dialog box.
- **Edit Comment.** Opens the Edit Comment dialog box for the selected item.
- **Delete Comment.** Deletes the selected comment.
- **Help.** Launches the online Help system and displays the topic for comments.

You can add as many comments as you wish to a source, either from the **Comments** window, or from the source itself.

TIP To add a comment to a specific object in the **Scene View**, use tags.

To toggle the Comments window

- Click **Review** tab ► **Comments** panel ► **View Comments**  .

 **Menu:** Classic user interface: View ► **Control Bars** ► Comments

To add a comment to a viewpoint

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** tool launcher  to open the Saved Viewpoints window.

- 2 Right-click the desired viewpoint in the Saved Viewpoints window, and click **Add Comment**.
- 3 In the **Comments** window, type in your comment. By default, it is assigned New status.
- 4 Click **OK**.

To add a comment to a viewpoint animation

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** tool launcher  to open the Saved Viewpoints window.
- 2 Right-click the desired viewpoint animation in the Saved Viewpoints window, and click **Add Comment**.
- 3 In the **Comments** window, type in your comment. By default, it is assigned New status.
- 4 Click **OK**.

To add a comment to a selection or search set

- 1 Click **Home** tab ► **Select & Search** panel ► **Sets** drop-down ► **Manage Sets** to open the **Sets** window.
- 2 Right-click the desired selection or search set, and click **Add Comment**.
- 3 In the **Comments** window, type in your comment. By default, it is assigned New status.
- 4 Click **OK**.

To add a comment to a clash result

- 1 Click **Home** tab ► **Tools** panel ► **Clash Detective**  to open the **Clash Detective** window, and click the **Results** tab.
- 2 In the **Results** area, right-click the desired clash result, and click **Add Comment**.
- 3 In the **Comments** window, type in your comment. By default, it is assigned New status.
- 4 Click **OK**.

NOTE Comments can also be assigned to a clash result when you assign a clash using the [Assign Clash Dialog Box](#) (page 741), and are automatically written when you unassign a clash, see [Unassign a Clash](#) (page 707).

To add a comment to a TimeLiner task

- 1 Click **Home** tab ► **Tools** panel ► **TimeLiner**  to open the **TimeLiner** window.
- 2 Click the **Tasks** tab.
- 3 Right-click the desired task, and click **Add Comment**.
- 4 In the **Comments** window, type in your comment. By default, it is assigned New status.
- 5 Click **OK**.

To view comments

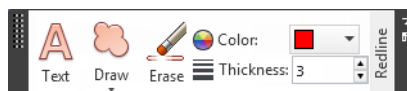
- 1 Open the **Comments** window.
- 2 Go the source of your comments. For example, open the **Saved Viewpoints** window.
- 3 As you click the source items, for example, viewpoints, the associated comments are displayed in the **Comments** window.

Icon	Description
	Selection set
	Search set
	Clash result (new)
	Clash result (approved)
	Clash result (reviewed)
	Clash result (resolved)
	Clash result (active)
	Viewpoint (orthographic camera)

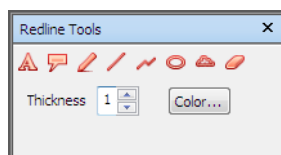
Icon	Description
	Viewpoint (perspective camera)
	Viewpoint animation
	Viewpoint animation cut
	Tag
	TimeLiner task (no attached items)
	TimeLiner task (with attached items)
	TimeLiner task (synchronized link)
	TimeLiner task (old or broken link)

Redline Tools Panel

The **Redline Tools** panel on the **Review** tab enables you to mark up viewpoints and clash results with redline annotations.



In the classic user interface, you can use the **Redline Tools** dockable window to add redlines, and tags.



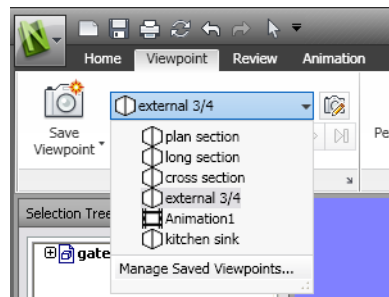
The **Thickness** and **Color** controls enable you to modify the redline settings. These changes do not affect already drawn redlines. Also, thickness only applies to lines; it does not affect redline text, which has a default size and weight and cannot be modified.

All redlines can only be added to a saved viewpoint or to a clash result which has a saved viewpoint. If you don't have any saved viewpoints, adding a tag will automatically create and save a viewpoint for you.

You can also convert measurements to redlines. For more information, see [Measuring](#) (page 393).

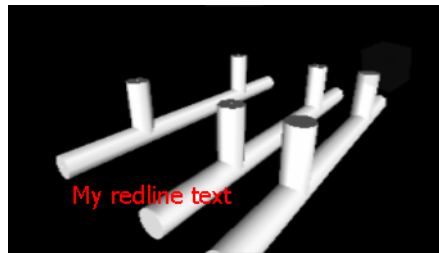
To add text

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



- 2 Click **Review** tab ► **Redline** panel ► **Text** .
- 3 In the **Scene View**, click the location where you want to place the text.
- 4 Enter your annotation into the box provided, and click OK. The redline is added to the selected viewpoint.

NOTE Text can only be added in a single line with this redline tool. To display text on multiple lines, write each line individually.

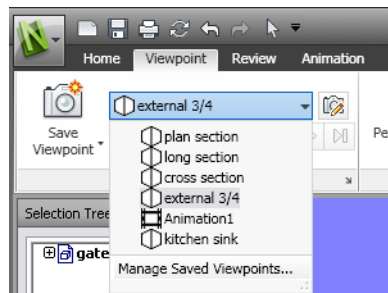


- 5 If you want to move the annotation, right-click the redline, and click **Move**. Clicking a different location in the **Scene View** moves the text there.
- 6 If you want to edit the annotation, right-click the redline, and click **Edit**.

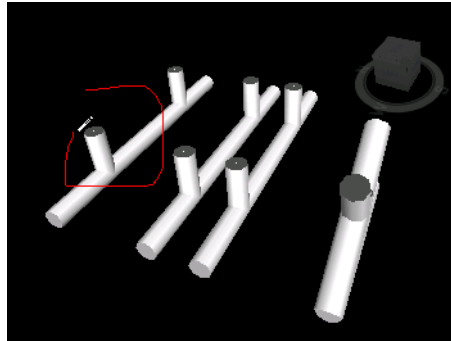
 **Menu:** Classic user interface: Review ► Redline ► Text

To draw freehand

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



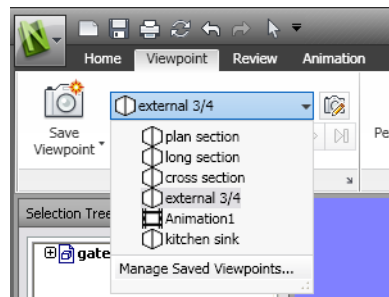
- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Freehand** .
- 3 Drag the mouse to draw in the viewpoint.



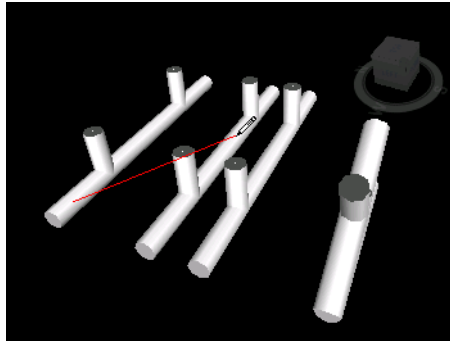
 **Menu:** Classic user interface: Review ► Redline ► Freehand

To draw a line

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



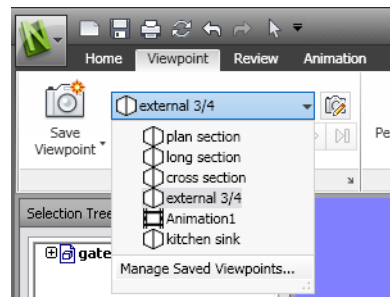
- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Line** .
- 3 Click in the viewpoint at line start and end points.



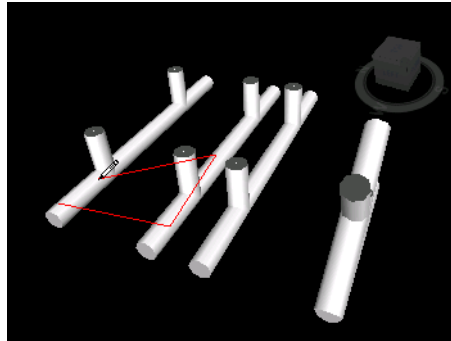
 **Menu:** Classic user interface: Review ► Redline ► Line

To draw a string of lines

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



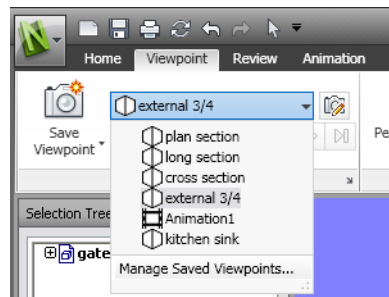
- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Line String**  .
- 3 Click in the viewpoint to begin. Each time you click, a new point is added to the line string. When the string is complete, click the right mouse button to end the line, and you can then start a new line string.



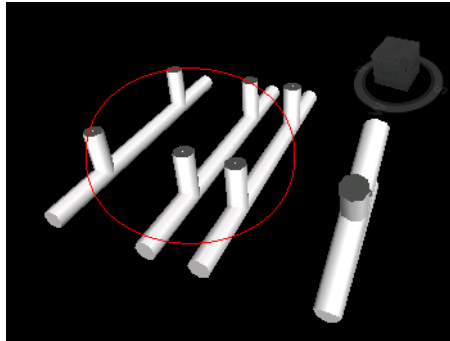
 **Menu:** Classic user interface: **Review** ► **Redline** ► **Line String**

To draw an ellipse

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



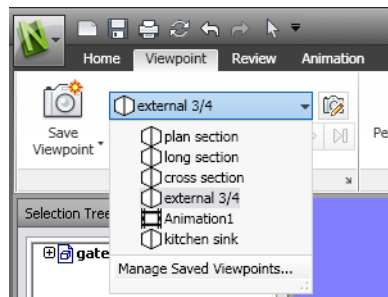
- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Ellipse** .
- 3 Click and drag a box in the viewpoint to outline the ellipse.
- 4 Release the mouse to place the ellipse in the viewpoint.



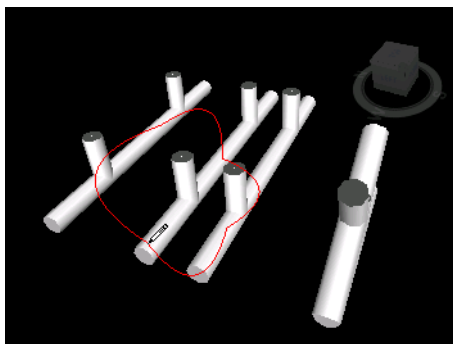
 **Menu:** Classic user interface: **Review** ► **Redline** ► **Ellipse**

To draw a cloud

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Cloud** .
- 3 Click in the viewpoint to begin drawing the arcs of the cloud. Each time you click, a new point is added. Clicking clockwise draws regular arcs, clicking counterclockwise, draws inverted arcs.
- 4 To automatically close the cloud, right-click.



 **Menu:** Classic user interface: **Review** ► **Redline** ► **Cloud**

Tool	Description
	Adds text to a viewpoint.
	Enables you to draw freehand in a viewpoint.
	Draws a line in a viewpoint.
	Draws a string of lines in a view point.
	Draws an ellipse in a viewpoint.
	Draws a cloud in a viewpoint.
	Erases redlines.

View Redlines and Tags

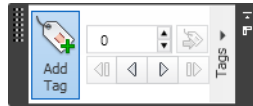
To view redlines and tags you need to recall the viewpoint that contains them.

To view redlines

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** tool launcher .
- 2 Click the desired viewpoint in the **Saved Viewpoints** window. All attached redlines (if any) are displayed in the **Scene View**.

Tags Panel

The **Tags** panel on the **Review** tab enables you to add and manage tags.

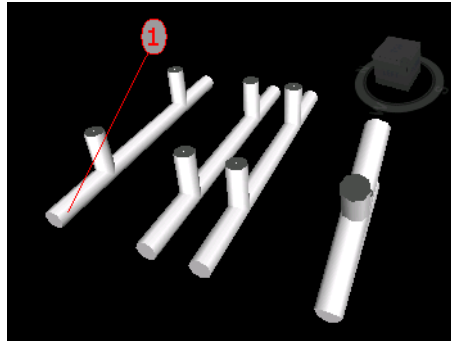


Tags combine the features of redlining, viewpoints and comments into a single, easy to use review tool. This allows you to tag anything you want to identify in the model scene. A viewpoint is automatically created for you, and you can add a comment and status to the tag.

For example, during a review session, you locate an item in the scene that is incorrectly sized or positioned. You can tag this item, stating the problem, save your review results as an NWF file, and pass the file to the design team. The design team can search the file, for any tags of status 'new', and locate your review comments. Once any necessary modifications are made to the drawing files, these can be reloaded into the *.nwf file, and the tag status can be changed accordingly. You can review this latest version of the NWF file, ensure all tags have been resolved and finally 'approve' them.

To add a tag

- 1 Click **Review** tab ► **Tags** panel ► **Add Tag**  .
 - 2 In the **Scene View**, click the object you want to tag.
 - 3 Click the area where you want the tag label to be located. The tag is now added, and both points are joined by a leader line.
- If the current viewpoint is not already saved, then it will be saved automatically and named "Tag View X", where "X" is the tag ID.



- 4 In the **Add Comments** dialog box, enter the text to be associated with the tag, set the tag **Status** from the drop-down list, and click OK.

 **Menu:** Classic user interface: **Review** ► **Redline** ► **Tag**

To view tags

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** tool launcher .
- 2 Click the desired viewpoint in the **Saved Viewpoints** window. All attached tags (if any) are displayed in the **Scene View**.

Edit Comments and Tags

Once a comment is saved, you can edit it either from the **Comments** window, or from the source itself. Editing tags is the same as editing comments.

You can edit content, change status assigned to comments and tags, and delete comments and tags. You can also renumber tag and comments IDs, if necessary (see [Manage Comment and Tag IDs](#) (page 420)).

You can use the **Thickness** and **Color** controls on the **Review** tab ► **Redline** panel to modify the way tags will be drawn in the **Scene View**. These changes do not affect already drawn tags.

To change content and status of a comment or tag

- 1 View the comment or tag you want to edit in the **Comments** window.
- 2 Right-click the comment or tag, and click **Edit Comment**.
- 3 Modify the comment text as necessary.

- 4 Use the **Status** box to change the status.
- 5 Click **OK**.

 **Menu:** Classic user interface: **Review** ► **Comments** ► **Edit Comment**

To delete a comment or tag

- 1 View the comment or tag you want to delete in the **Comments** window.
- 2 Right-click the comment or tag, and click **Delete Comment**.

 **Menu:** Classic user interface: **Review** ► **Comments** ► **Delete Comment**

Edit Redlines

Added redlines cannot be edited with the exception of text, which can be moved and changed as needed.

To move text

- Right-click the annotation in the **Scene View**, and click **Move** on the shortcut menu.

To edit text

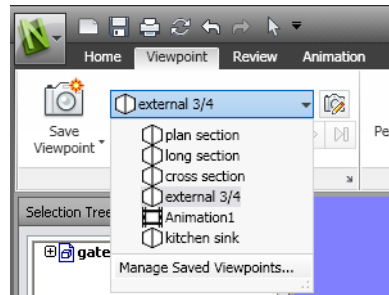
- 1 Right-click the annotation in the **Scene View**, and click **Edit** on the shortcut menu.
- 2 Enter a new annotation in the box provided, and click **OK**.

To delete text

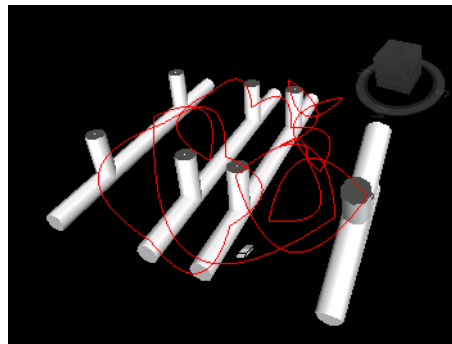
- Right-click the text you want to delete, and click **Delete Redline** on the shortcut menu.

To erase redlines

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** drop-down, and choose the viewpoint that you want to review.



- 2 Click **Review** tab ► **Redline** panel ► **Draw** drop-down, and click **Erase** .
- 3 Drag a box over the redline you want to delete, and release the mouse.



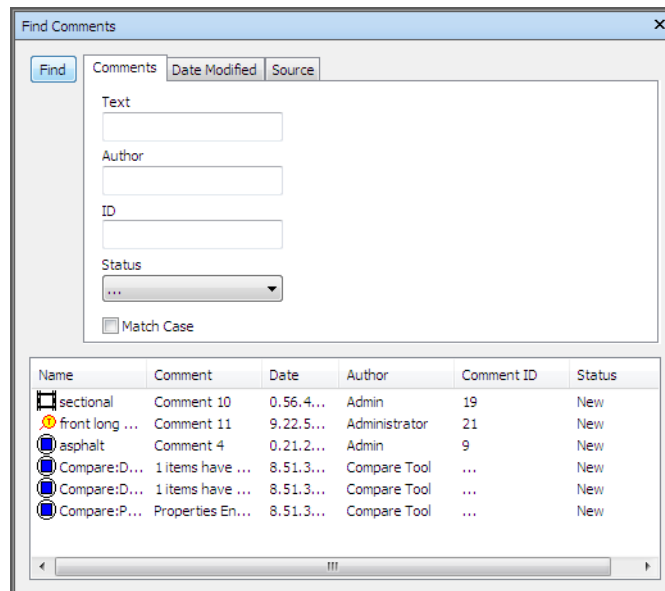
 **Menu:** Classic user interface: **Review** ► **Redline** ► **Erase**

Find Comments and Tags

You can use the **Find Comments** window to set up and run a search for tags and comments. You can also use the controls on the **Tags** panel to navigate through tags.

Find Comments Window

The **Find Comments** window is a dockable window that enables you to search for comments and tags based on the comment data (text, author, comment ID, status), comment modification date, and the source of comment.



Three tabs on the top enables you to set up the search criteria. Pressing the **Find** button runs the search. Any found results are displayed in a multi-column table at the bottom of the window. You can use the scroll bars at the right and at the bottom of the tab to move through the comments. There are different icons helping you to identify the source of each comment at a glance. These icons are the same as the ones used in the **Comments** window.

Selecting a comment in the list also selects the source of the comment. For example, selecting a comment which originated from a saved viewpoint selects that viewpoint.

You can change the sort order of the items in the list by clicking a column head.

If new comments are added, or any of the existing comments are modified, the results list is cleared.

To toggle the Find Comments window

■ Click **Review** tab ➤ **Comments** panel ➤ **Find Comments** .

 **Menu:** Classic user interface: Edit ➤ Find Comments or Review ➤ Comments ➤ Find Comments

 **Command entry:** SHIFT + F4

To find comments

- 1 Open the **Find Comments** window.
- 2 Set up the search criteria:
 - Use the **Comments** tab if you want to search for a specific text, author, comment ID, or status.
 - Use the **Date Modified** tab to only search for comments created within the specified timeframe.
 - Use the **Source** tab to only search for comments attached to the selected sources.
- 3 Click **Find**.

To find tags by using the Find Comments window

- 1 Open the **Find Comments** window.
- 2 Click the **Source** tab, select the **Redline Tags** check box, and clear the rest of the check boxes.
- 3 Use the **Comments** and **Date Modified** tabs to restrict your search further, if required.
- 4 Click **Find**.

Comments Tab

Use this tab to restrict your search based on the comment data. If the boxes on this tab are left empty, the search returns all comments within the criteria set on the **Date Modified** and **Source** tabs.

Text The exact text to search for in all comments. This can be a word or a few words. So for, example, typing *redline*, and running a search, returns only the comments that contain a single “redline” word.

If you don’t know the exact content of the comment, you can use the wildcards. To match one single unspecified character use the symbol “?” (question mark). To match any number of unspecified characters either before or after the text, use the symbol * (asterisk). For example, if you want to find all comments that contain some text before and after the word *redline*, type **redline** in the **Text** box.

Author The exact author name to search for in all comments. Use the wildcards if you don’t know the exact author name.

ID The exact comment ID to search for. You can only use numbers here.

Status Selects the comment status to search for.

Match Case Select this check box if you want the search to respect lower and upper case characters in the search.

Date Modified Tab

Use this tab to specify a date range within which the comments must have been made.

All Comments The search returns all comments. This is the default option.

Between Select this radio button if you want to search for all comments modified between the specified dates.

During the Previous Months Select this radio button to locate all comments modified in the last X number of months.

During the Previous Days Select this radio button to locate all comments modified in the last X number of days.

Source Tab

Use this tab to restrict the search by the source that the comment is attached to. By default all check boxes are selected.

Clash Detective Select this check box if you want to find the comments attached to Clash Detective results. Clearing this check box excludes the comments related to clashes from the search results.

TimeLiner Select this check box if you want to find the comments attached to **TimeLiner** tasks. Clearing this check box excludes the comments related to **TimeLiner** from the search results.

Viewpoints Select this check box if you want to find the comments attached to viewpoints. Clearing this check box excludes the comments related to viewpoints from the search results.

Redline Tags Select this check box if you want to find the comments attached to tags. Clearing this check box excludes the comments related to tags from the search results.

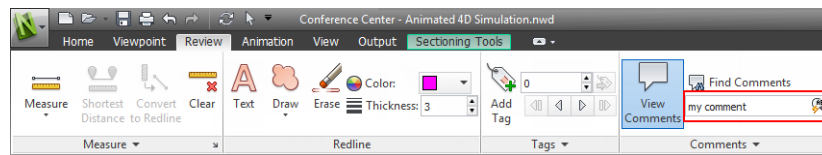
Sets Select this check box if you want to find the comments attached to selection and search sets. Clearing this check box excludes the comments related to selection and search sets from the search results.

Quick Find Comments

To locate comments that match a specified search term quickly, use the **Quick Find Comments** feature.

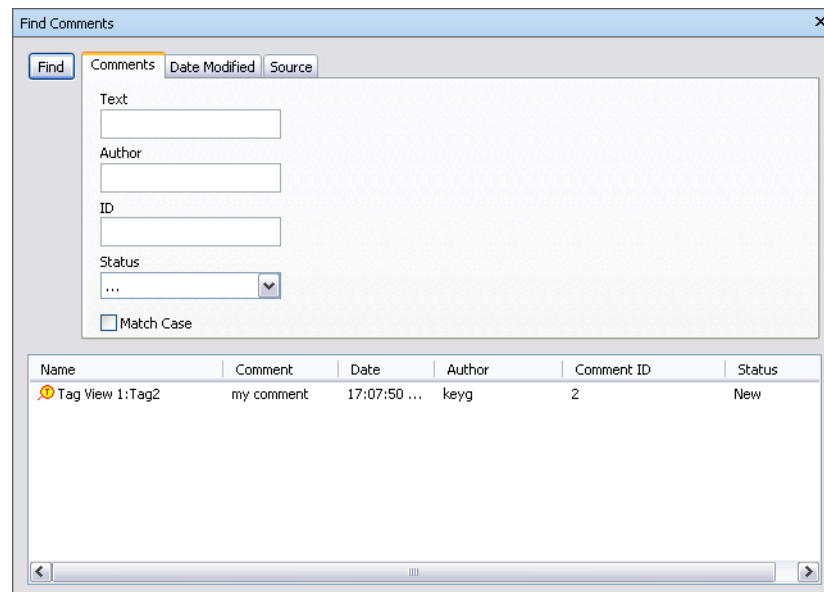
To quickly find comments

- 1 Click **Review** tab ► **Comments** panel.
- 2 In the **Quick Find Comments** text box, type in the string to search for in all comments. This can be a word or a few words.



- 3 Click **Quick Find Comments** .

The **Find Comments** window opens with a list of all comments that match the entered text.



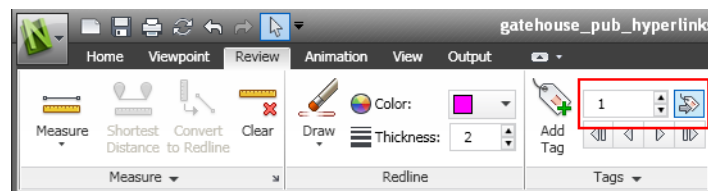
Clicking on the comment in the list takes you to the appropriate viewpoint.

Find Tags

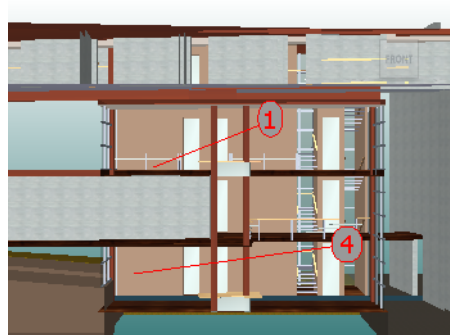
You can search for tags the same way as you search for comments. You can also find tags by their ID numbers, and navigate through tags using the controls on the **Tags** panel.

To find tags by tag ID

- 1 Click **Review** tab ► **Tags** panel.
- 2 Enter the tag ID into the text box, and click **Go to Tag** .



You are automatically taken to the appropriate viewpoint.



To navigate through tags

- 1 Click **Review** tab ► **Comments** panel ► **View Comments**  to open the **Comments** window.
- 2 Click **Review** tab ► **Tags** panel ► **First Tag**  . The tag comment is shown in the **Comments** window, and the **Scene View** displays the viewpoint with the first tag.

- 3 To navigate through the tags in your scene:
 - Click **Review** tab ► **Tags** panel ► **Next Tag** ▸ to find the tag following the current tag.
 - Click **Review** tab ► **Tags** panel ► **Previous Tag** ◀ to find the tag preceding the current tag.
 - Click **Review** tab ► **Tags** panel ► **Last Tag** ⏮ to find the last tag in the scene.

Manage Comment and Tag IDs

When you add a tag or comment to your scene, it is automatically assigned a unique ID. If, however, you are appending or merging multiple Autodesk Navisworks files together, then there is a possibility for the same ID to be used more than once. Consider the following example. Three users reviewed and added comments and redlines to same model file, each saving their work as an NWF file. If you merge the resulting NWF files together, only one copy of the geometry would be loaded and any tag viewpoints of the same name would be suffixed with the NWF filename in brackets. All tag IDs, however, would be retained. In such cases, you can renumber all the IDs, making them unique to the scene once again.

NOTE There may be a situation, where two sessions are merged that contain identically numbered tags and corresponding viewpoints (Tag Views). In this situation, when you renumber tag IDs, Autodesk Navisworks will also attempt, wherever possible, to rename the associated Tag Views in line with the new tag numbers.

To renumber comment IDs

- 1 Click the **Review** tab, and slide out the **Comments** panel.
- 2 Click **Renumber Comment IDs** .

To renumber tag IDs

- 1 Click the **Review** tab, and slide out the **Tags** panel.
- 2 Click **Renumber Tag IDs** .

Links

There are several sources of links in Autodesk Navisworks: original links that have been converted from the native CAD files, links that have been added by Autodesk Navisworks users, and links that have been automatically generated by the program (for example, selection set links, viewpoint links, **TimeLiner** task links and so on).

The links converted from the native CAD files, and the links added by Autodesk Navisworks users are treated as object properties. This means, you can examine them in the **Properties** window.

You can also use the **Find Items** window to search for them.

All links are saved with Autodesk Navisworks files so that as the model changes, the links remain there for you and others to view.

Link Categories

There are two types of links: standard and user-defined.

Standard links are split into the following categories:

- Hyperlink
- Label
- Viewpoints
- Clash Detective
- **TimeLiner**
- Sets
- Redline tags

By default, all links except labels, are drawn as icons in the **Scene View**. Labels are drawn as text.

User-defined links enable you to customize the link categories to suit your workflow. By default, user-defined links are drawn as icons in the **Scene View**.

You can use the **Options Editor** to toggle the display of each of the link categories, and also to control their appearance.

When you add a link, you can assign a user-defined category, hyperlink category, or label category to it. The rest of the categories are automatically assigned by Autodesk Navisworks when it generates corresponding links.

Display Links

You can switch links in the **Scene View** on and off. You can also toggle the display of each of the link categories. Autodesk Navisworks remembers the selected visibility setting between sessions.

When links are switched on, you can reduce the screen clutter by restricting a number of links can be shown in the **Scene View**, hiding colliding icons, and using culling. Finally, as some standard link categories can have comments associated with them, you can choose to only draw links with attached comments.

To toggle the display of links

■ Click **Home** tab ► **Display** panel ► **Links** .

 **Menu:** Classic user interface: **Tools** ► **Links**

To control the display of standard links

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, expand the **Links** node, and click the **Standard Categories** option.
- 3 On the **Standard Categories** page, select the **Visible** check box to display the corresponding link category. Clearing the check box hides the corresponding link category in the **Scene View**.
By default, all standard link categories are visible.
- 4 Click **OK**.

To control the display of user-defined links

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, expand the **Links** node, and click the **User-Defined Categories** option.
- 3 On the **User-Defined Categories** page, select the **Visible** check box to display the corresponding link category. Clearing the check box hides the corresponding link category in the **Scene View**.
By default, all user-defined link categories are visible.

NOTE If no user-defined categories have been added, this page is empty.

- 4 Click **OK**.

To reduce the screen clutter

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Links** option.
- 3 On the **Links** page, enter the number of links into the **Max Icons** box. By default, 25 links can be visible.
- 4 To hide links that appear overlapped in the **Scene View**, select the **Hide Colliding Icons** check box.
- 5 In the **Cull Radius** box enter the desired value. Only the links located within the specified distance from the camera are drawn in the **Scene View**. The default value of 0 means that all links are drawn.
- 6 Click **OK**.

To hide links without comments

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, expand the **Links** node, and click the **Standard Categories** option.
- 3 On the **Standard Categories** page, select the **Hide Icons Without Comments** check box for all required link categories. By default, links without comments are also displayed.
- 4 Click **OK**.

Customize Links

You can customize the default appearance of links in Autodesk Navisworks. In particular, you can draw them in 3D, and you can add leader lines (arrows) pointing to the attachment point on the items. You can also choose how to represent each link category (as an icon or as text).

Attachment Points

By default, links are attached to the default center of the item's bounding box.

You can override this with more convenient attachment points. If you add more than one attachment point, the link is displayed attached to the closest attachment point to the camera during navigation. This allows you to set up links so that they are always available for following when drawn in 3D mode during navigation, and do not disappear behind objects.

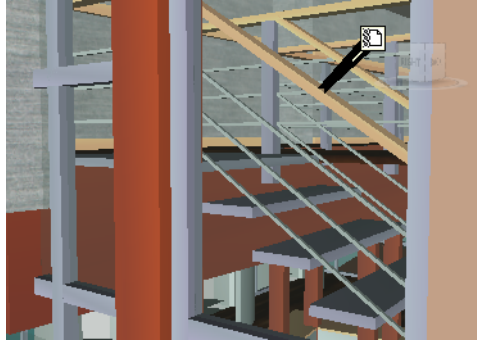
To draw links in 3D mode

Note: In 3D mode links can become hidden by other objects in the scene when you are navigating.

- 1 Click the application button  **> Options.**
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Links** option.
- 3 On the **Links** page, select the **In 3D** check box.
Links now float in 3D space just in front of their attachment points to the items.
- 4 Click **OK**.

To show leader lines

- 1 Click the application button  **> Options.**
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Links** option.
- 3 On the **Links** page, Enter the X- and Y- distance in **Leader Offset** for the number of pixels to the right and up that these leader lines will use. The default angle is 0. The recommended angle is 45.
Links in the **Scene View** have now leader lines pointing to the attachment point on the items.



- 4 Click **OK**.

To customize appearance of standard links

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, expand the **Links** node, and click the **Standard Categories** option.
- 3 On the **Standard Categories** page, use the **Icon Type** box to specify how you want a link to be drawn for each of the available categories. You can choose between an icon and text.
By default, label links are shown as text, and the rest of the link categories are shown as icons.
- 4 Click **OK**.

To customize appearance of user-defined links

- 1 Click the application button  **> Options**.
 - 2 In the **Options Editor**, expand the **Interface** node, expand the **Links** node, and click the **User-Defined Categories** option.
 - 3 On the **User-Defined Categories** page, use the **Icon Type** box to specify how you want a link to be drawn for each of the available categories. You can choose between an icon and text. By default, links with user-defined categories are shown as icons.
- NOTE** If no user-defined categories have been added, this page is empty.
- 4 Click **OK**.

The table below shows the icons that can be used to represent different link categories in the **Scene View**.

Icon	Description
	Represents links that have hyperlink, label, or any user-defined category (and points to a web address).
	Represents links that have hyperlink, label, or any user-defined category (and points to an external file).
	Represent links with Clash Detective category (new clashes).
	Represent links with Clash Detective category (active clashes).
	Represent links with Clash Detective category (reviewed clashes).
	Represent links with Clash Detective category (resolved clashes).
	Represent links with Clash Detective category (approved clashes).
	Represent links with TimeLiner category (task with attached items).
	Represent links with TimeLiner category (task with valid links).
	Represent links with TimeLiner category (tasks with broken links).
	Represents links with viewpoints category (perspective camera mode).
	Represents links with viewpoints category (orthographic camera mode).
	Represents links with tags category.
	Represents links with sets category (selection sets).

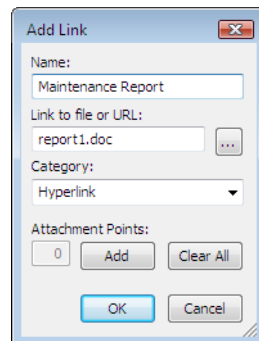
Icon	Description
	Represents links with sets category (search sets).

Add Links

You can add links that point to various data sources, such as spreadsheets, webpages, scripts, graphics, audio and video files and so on. An object can have multiple links attached to it, although, only one link, called default, is shown in the **Scene View**. The default link is the link that you add first, but you can mark a different link as default, if necessary.

To add a link to an object

- 1 In the **Scene View**, select the desired geometry item.
- 2 Click **Item Tools** tab ► **Links** panel ► **Add Link** .
- 3 In the **Add Link** dialog box, enter the name for the link in the **Name** box.
- 4 In the **Link to File or URL** box, type in the full path to the required data source or the URL address. You can also browse  to the folder containing the desired external file.
- 5 Choose the category for your link from the **Category** drop-down list. By default, your link has hyperlink category.



TIP To create a custom category type, type its name directly into the **Category** box. When you save your link, the corresponding user-defined category is automatically created.

- 6 Optional: By default, your link is attached to the default center of the item's bounding box. If you want to attach your link to a specific point on the selected item, click the **Add** button. A cross-hair cursor appears in the **Scene View**, allowing you to select a point on the item where the link will be attached to.

NOTE If you made a mistake, clicking the **Clear All** button deletes all attachment points associated with this link and reverts to the link being attached to the center of the item's bounding box.

- 7 Click **OK**.

To add several links to the same object

- 1 In the **Scene View**, select the desired geometry item.
- 2 Right-click and click **Links ► Add Link**.
- 3 Use the **Add Link** dialog box to add the first link. This is the default link, and it's the only link that will be visible in the **Scene View**. You can choose a different default link later, if necessary.
- 4 Right-click the object again, and repeat the previous steps to add all required links.

Find and Follow Links

Links are an extremely useful review tool to allow you to access non-graphical information through the graphical interface of Autodesk Navisworks.

The links converted from the native CAD files, and the links added by Autodesk Navisworks users are treated as object properties. This means, you can examine them in the **Properties** window.

You can also use the **Find Items** window to search for them.

To follow a default link

- 1 Make sure links are switched on. If not, click **Home** tab ► **Display** panel ► **Links**  .
- 2 Click the desired link in the **Scene View** to open the attached data source.

Shortcut menu: Follow Link

To follow one of the non-default links

- 1 Make sure links are switched on. If not, click **Home** tab ► **Display** panel ► **Links**  .
- 2 Right-click the default link, and click the **Select Item Containing Link** option. This option is only available for multiple links attached to the same item.
- 3 Right-click the selected item, and then click **Links** ► the link that you want to follow.

To follow a default link from the Properties window

Note: You can use this procedure to follow links from original CAD files, or links that you have added yourself.

- 1 In the **Scene View**, select the object with a link.
- 2 Open the **Properties** window, right-click any tab, and click **Follow Default Link**.

To find links

- 1 Click **Home** tab ► **Select & Search** panel ► **Find** Items  .
- 2 In the **Find Items** window, click the **Standard** tab, press and hold CTRL, and click all files that comprise the model.
- 3 Click the **Category** column, and select **Hyperlinks** from the drop-down list.
- 4 In the **Property** column, select the property name from the drop-down list, for example, "Name".
- 5 In the **Condition** column, select the condition operator, for example, '≡'.
- 6 In the **Value** column, type in the property value to search for, for example, "My Link".
- 7 Click the **Find All** button. The search results are highlighted in the **Scene View** and the **Selection Tree**.

Manage Links

You can edit original links that have been converted from the native CAD files. If you do this, then save the changes in an NWF file, then change the link in the original CAD file, and reopen the NWF file in Autodesk Navisworks, then your edit “overrides” will remain. If you haven’t edited the links in Autodesk Navisworks, however, the updated links from the CAD file will appear.

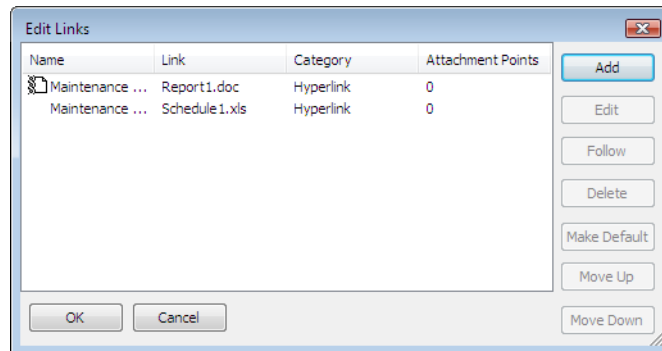
You can reset all links on an object to those that were originally converted from the CAD file. You can also reset all links on all objects in the scene to their original state.

NOTE You cannot edit automatically generated links, such as the links that point to clash results, **TimeLiner** tasks and so on.

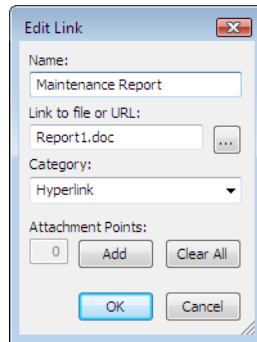
You can delete any links that have been added to your file by Autodesk Navisworks users, and the links that have been converted from original CAD files. You cannot delete automatically generated links, but you can turn them off, so they don’t clutter the **Scene View**.

To edit a link

- 1 In the **Scene View**, right-click the link you want to edit, and click **Edit Link**.
- 2 In the **Edit Links** dialog box, click the link you want to change, and click the **Edit** button.



- 3 Use the **Edit Link** dialog box to modify the link details, and click **OK**.



- 4 Click OK to close the **Edit Links** dialog box.

Ribbon: **Item Tools** tab ► **Links** tab ► **Edit Links** 

To change the default link

- 1 In the **Scene View**, right-click the default link, and click **Edit Link**.
- 2 In the **Edit Link** dialog box, select the link that you want to be shown in the **Scene View**, and click the **Make Default** button.

NOTE You can also move links up and down the list using the **Move Up** and **Move Down** buttons, or by dragging them to their new position in the list. The link at the top of the list is the default link.

- 3 Click **OK**.

To delete a link

- 1 In the **Scene View**, right-click the default link, and click **Edit Link**.
- 2 In the **Edit Links** dialog box, click the link you want to delete, and click the **Delete** button.
- 3 Click **OK**.

 **Menu:** Classic user interface: **Review** ► **Links** ► **Edit Links**

To reset all links for an object

Attention: Resetting links for an object also removes any links manually added to it by Autodesk Navisworks users. If you made a mistake, use the **Undo** ↶ button on the **Quick Access** toolbar.

- 1 In the **Scene View**, select the object with the links that you want to reset to their original state.
- 2 Click **Item Tools** tab ► **Links** panel ► **Reset Links** 🔗.

To reset all links in a scene

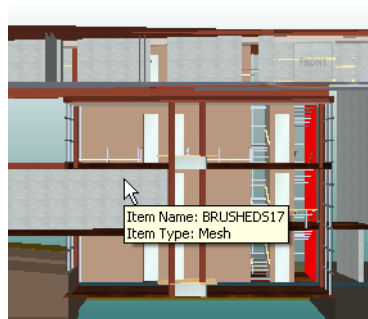
Attention: Resetting links in a scene also removes all links manually added to it by Autodesk Navisworks users. If you made a mistake, use the **Undo** ↶ button on the **Quick Access** toolbar.

- Click **Home** tab ► **Project** panel ► **Reset All** drop-down ► **Links** 📄.

Quick Properties

You can switch quick properties in the **Scene View** on and off. Autodesk Navisworks remembers the selected visibility setting between sessions.

When **Quick Properties** are switched on, you can view property information in a tooltip style window as you move your cursor over objects in the **Scene View**. You don't need to select objects first. The quick properties tooltip disappears after a few seconds.



By default, quick properties show the name and type of the object, but you can use the **Options Editor** to define which properties are shown. Each

definition that you configure enables you to display an additional category/property combination in quick properties. You can choose whether to hide category names in quick properties or not.

NOTE When you move your mouse over an object that doesn't have the requested property, Autodesk Navisworks searches up the selection tree for a parent object that contains that information, and displays it instead, thus maximizing the useful information you get.

To toggle the display of quick properties

■ Click **Home** tab ► **Display** panel ► **Quick Properties** .

 **Menu:** Classic user interface: **Tools** ► **Quick Properties**

To add quick properties definition

- 1 Click the application button  ► **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, expand the **Quick Properties** node, and click the **Definitions** option.
- 3 On the **Definitions** page, click **Grid View**  to display quick properties definitions as table rows.
- 4 Click **Add Element** . A new row is added to the top of the table.
- 5 Click the **Category** column, and select the property category from the drop-down list, for example 'Item.' The options available depend on the property categories in your model.
- 6 Click the **Property** column, and select the property name from the drop-down list, for example, 'Material'. The options available depend on the selected property category.
- 7 Click **OK**.

NOTE You can add as many definitions to your quick properties as you like.

To delete quick properties definition

- 1 Click the application button  ► **Options**.

- 2 In the **Options Editor**, expand the **Interface** node, expand the **Quick Properties** node, and click the **Definitions** option.
- 3 On the **Definitions** page, click **Grid View**  to display quick properties definitions as table rows.
- 4 Click the **Category** or **Property** for the definition that you want to delete.
- 5 Click **Remove Element** .
- 6 Click **OK**.

To hide category names

- 1 Click the application button  **> Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Quick Properties** option.
- 3 Select the **Hide Category** check box.
- 4 Click **OK**.

SwitchBack

SwitchBack enables you to select an object in Autodesk Navisworks, and then locate and zoom into the same object in a native CAD package. You can use the **SwitchBack** functionality with AutoCAD (version 2004 or later), Revit (version 2012 or later), and MicroStation-based CAD products (/J and v8).

IMPORTANT The native CAD package must be installed on the same machine as Autodesk Navisworks for **SwitchBack** to work.

To use **SwitchBack** with AutoCAD

- 1 For AutoCAD (version 2004 or later) or products based on it, first open the product in the usual manner, and type: `nwload` in the command line to load the `nwexport` plugin.

NOTE If SwitchBack needs to be available every time AutoCAD is run, `nwexport` can be added to the set of startup applications in AutoCAD.

- 2 Once the CAD package is running, and nwexport has been started, return to Autodesk Navisworks.

- 3 Click the application button  ► **Options**.
- 4 In the **Options Editor**, expand the **File Readers** node, and click the **DWG/DXF** option.
- 5 On the DWG/DXF page, check that the **Convert Entity Handles** option is selected.
- 6 Select an object in the **Scene View**, and click **Item Tools** tab
► **SwitchBack** panel ► **SwitchBack** .

The current Autodesk Navisworks camera view is taken back to the CAD package, and the same object is selected. Selection of objects is done by entity handle.

TIP Alternatively, in the **Clash Detective** window, on the **Results** tab, you can click the **SwitchBack** button.

- 7 Make the changes in AutoCAD, then save the changes.
- 8 Return to Autodesk Navisworks, and click **Refresh**  on the **Quick Access** toolbar to view the modified object.

NOTE Some objects cannot be selected in AutoCAD (for example, blocks) which may mean that running SwitchBack may be unsuccessful with a given selected object. If this is the case, try selecting further up the object tree and trying again.

Shortcut menu: SwitchBack

To use SwitchBack with MicroStation

- 1 For MicroStation (/J and v8) or products based on it, open the product in the usual manner.
- 2 Click **Utilities** ► **Key In**.
- 3 In the **Key-In** dialog box, type `mdl load nwexport9` to load the nwexport plugin.

TIP If **SwitchBack** needs to be available every time MicroStation is run, `nwexport9` can be added to the list of MDL plugins in MicroStation.

- 4 Once the MicroStation package is running, and nwexport has been started, return to Autodesk Navisworks.

- 5 Select an object in the **Scene View**, and click **Item Tools** tab

➤ **SwitchBack** panel ➤ **SwitchBack** .

The current Autodesk Navisworks camera view is taken back to first visible view in MicroStation, and the same object is selected. Selection of objects is done by element ID (MicroStation v8) or DMRS value (MicroStation /J).

TIP Alternatively, in the **Clash Detect** window, on the **Results** tab, you can click the **SwitchBack** button.

- 6 If you want to override the view used by SwitchBack, use the `nwview <view number> key-in` where view number is the visible window in MicroStation. This view setting is not saved between sessions.
- 7 Make the changes in MicroStation, then save the changes.
- 8 Return to Autodesk Navisworks, and click **Refresh**  on the **Quick Access** toolbar to view the modified object.

Shortcut menu: SwitchBack

To use SwitchBack with Revit

- 1 For Revit 2012 or products based on it, open the product in the usual manner, and initialize the **Navisworks SwitchBack** add-in:
 - 1 Open any existing project, or create a new one.
 - 2 Click **Add-Ins** tab ➤ **External Tools** ➤ **Navisworks SwitchBack** to enable it. You can now close the project, but don't close Revit.
- 2 Return to Autodesk Navisworks and open the desired file. As long as you are working with an NWC file exported from Revit, or a saved NWF or NWD file, you can SwitchBack to Revit.
- 3 Select an object in the **Scene View**, and click **Item Tools** tab

➤ **SwitchBack** panel ➤ **SwitchBack** .

Revit will load the relevant project, find and select the item, and zoom to it. If **SwitchBack** was unsuccessful with the selected object, and you receive an error message, try selecting further up the **Selection Tree** in Autodesk Navisworks.

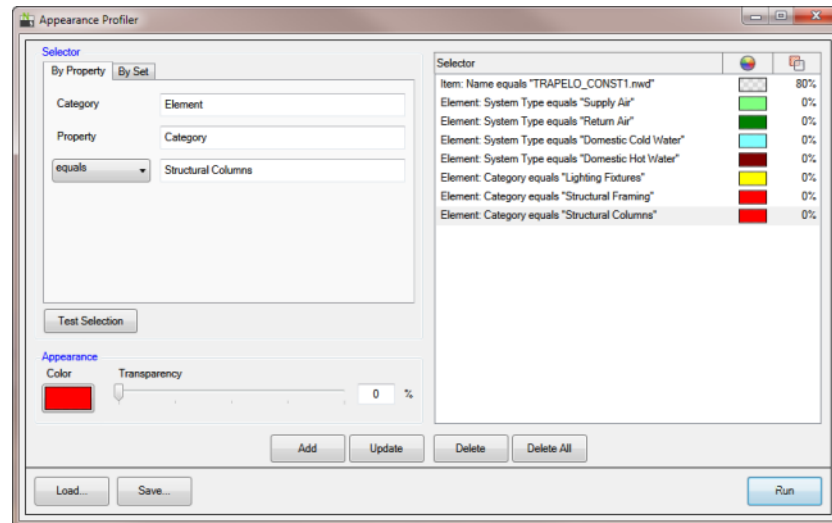
TIP Alternatively, in the **Clash Detect** window, on the **Results** tab, you can click the **SwitchBack** button.

NOTE If you try to use **SwitchBack** and the RVT file is not in the same location as when it was saved, a dialog box will appear asking you to browse to the RVT file.

Shortcut menu: SwitchBack

Appearance Profiler

The Appearance Profiler allows you to set up custom appearance profiles based on sets (search and selection) and property values, and use them to color-code objects in the model to differentiate system types and visually identify their status. Appearance profiles can be saved as DAT files and shared between other Autodesk Navisworks users.



Appearance profile selectors define the object selection criteria and appearance settings. The object selection can be based on property values or on search and selection sets available in your Autodesk Navisworks file.

Using property values is more flexible, because search and selection sets need to be added to your model first, and are frequently designed to cover a specific area of the model (level, floor, zone and so on). For example, if your model has five floors, to locate all 'Cold Water' objects with sets, you will need to set up five 'Cold Water' selectors - one per floor. If you use a property-based method, one 'Cold Water' selector is sufficient, as the search encompasses all

of the model, including extra properties coming from external databases if available (see [External Database Links](#) (page 371)).

There is no restriction as to how many selectors an appearance profile can have. However, the order of selectors within a profile is important. The appearance selectors are applied to the model sequentially, from top to bottom. If an object belongs to more than one selector, the object's appearance will be overridden each time it is processed by a new selector in the list. Currently you cannot change the order of selectors once you added them to the list.

See also:

[Appearance Profiler Dialog Box](#) (page 739)

To open the Appearance Profiler

■ Click **Home** tab ➤ **Tools** panel ➤ **Appearance Profiler** 

To save an appearance profile

- 1 Open the **Appearance Profile** dialog and define all required sectors.
- 2 Click the **Save button**.
- 3 In the **Save As** dialog, enter a file name and location.
- 4 Click **Save**.

To open an existing appearance profile

- 1 Open the **Appearance Profile** dialog.
- 2 Click **Load**.
- 3 In the **Open** dialog, locate the desired appearance profile DAT file.
- 4 Click **Open**.

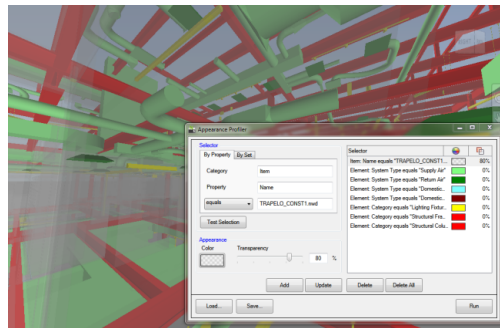
To color-code model by property values

- 1 Open the **Appearance Profile** dialog.
- 2 Click the **By Property tab** in the **Selector** area.
- 3 Use the fields provided to configure the object selection criteria for your selector.
- 4 Click **Test Selection**. All objects that satisfy the criteria are selected in the **Scene View**.

- 5 If you are satisfied with the results, use the **Appearance** area to configure the color and transparency overrides for your selector.
- 6 Click **Add**. The selector is now added to the **Selector** list.
- 7 Repeat steps 3 - 6 until you have added all required selectors. Remember, the order of selectors in the list is important.

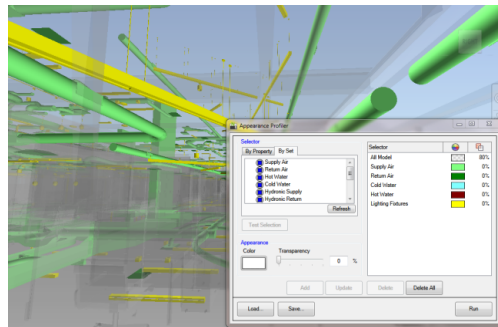
TIP If you use your first selector to override the color of the entire model to gray with 80% transparency, other color overrides will stand out more.

- 8 Click **Run**. The objects in your model are now color-coded.



To color-code model by search and selection sets

- 1 Open the **Appearance Profile** dialog.
- 2 Click the **By Set** tab in the **Selector** area.
- 3 Select the set you want to use in the list, and click **Test Selection**. All objects that satisfy the criteria are selected in the **Scene View**.
- 4 If you are satisfied with the results, use the **Appearance** area to configure the color and transparency overrides for your selector.
- 5 Click **Add**. The selector is now added to the **Selector** list.
- 6 Repeat steps 3 - 5 until you have added all required selectors. Remember, the order of selectors in the list is important.
- 7 Click **Run**. The objects in your model are now color-coded.



To reset color overrides back to original values

- Click **Home** tab ► **Project** panel ► **Reset All drop-down** ► **Appearances**

Use Viewpoints and Sectioning Modes

8

Viewpoints are an important feature of Autodesk Navisworks. They allow you to save and recall different settings related to the view of the model and settings for navigation. You also have the option to save item visibility and appearance overrides within viewpoints.

Create and Modify Viewpoints

Overview of Viewpoints

Viewpoints are snapshots taken of the model as it is displayed in the **Scene View**. Importantly, viewpoints can be used for more than just saving information about the view of the model. For example, they can be annotated with redlines and comments, allowing you to use viewpoints as a design review audit trail. Viewpoints can also be used as links in the **Scene View**, so that when you click on and zoom to the viewpoint, Autodesk Navisworks also displays the redlines and comments associated with it.

The viewpoints, redlines and comments are all saved into an NWF file from Autodesk Navisworks, and are independent of the model geometry. So, if the native CAD files are changing, the saved viewpoints remain the same, appearing as an overlay on top of the base layer of model geometry. This enables you to see how the design has evolved. See [Review Your Model](#) (page 337) for more information on links, comments and redlines and [Native File Formats](#) (page 163) for more information on the NWF file format.

Viewpoints encompass a range of different information about the view of the model, navigation settings, and annotations in the form of redlines and comments. See [Default Viewpoint Options](#) (page 451) for more information.

View of the Model

- Camera position, projection mode, field of view and orientation
- Lighting mode, render mode and toggles for the display of different geometry types (surfaces, lines, points)
- Sectioning configuration

Additionally, the following item overrides can be saved with the viewpoint (this is optional):

- Visibility (hidden / required)
- Appearance (color and transparency)

Navigation

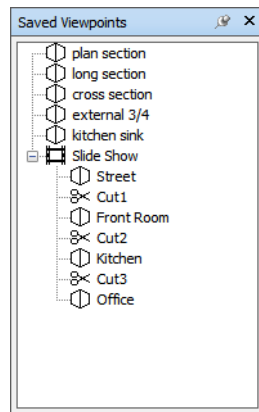
- Linear and angular speeds of motion
- Realism settings (collision, gravity, third person, crouch)
- The currently selected navigation tool

Annotations

- Redlines
- Comments

Saved Viewpoints Window

The **Saved Viewpoints** window is a dockable window that enables you to create and manage different views of your model so that you can jump to preset viewpoints without having to navigate each time to reach an item.



Viewpoint animations are also saved with the viewpoints, as they are simply a list of viewpoints treated as keyframes. In fact, viewpoint animations can be made by simply dragging preset viewpoints onto an empty viewpoint animation. You can organize your viewpoints and viewpoint animations using folders.

Icons are used to represent different elements:

 represents a folder which may contain all other elements (including other folders).

 represents a viewpoint saved in orthographic mode.

 represents a viewpoint saved in perspective mode.

 represents a viewpoint animation clip.

 represents a cut inserted into a viewpoint animation clip.

You can select more than one viewpoint by either holding down the CTRL key and left-clicking, or by left-clicking the first item, and then clicking the last item while holding down the SHIFT key.

You can drag viewpoints around the **Saved Viewpoints** window, and reorganize them into folders or animations.

There are no buttons on this window, and commands are invoked through shortcut menus.

Through these menus, you can save and update viewpoints, create and manage viewpoint animations, and create folders to organize these viewpoints and viewpoint animations. You can also drag and drop viewpoints or viewpoint

animations onto viewpoint animations or folders. Holding down the CTRL key during this operation will copy the element being dragged. This allows fairly complex hierarchies of viewpoint animations and folders to be easily composed.

Viewpoints, folders and viewpoint animations can all be renamed by slow clicking (clicking and pausing without moving the mouse) the element, or clicking it and pressing F2.

To toggle the Saved Viewpoints window

- Click **Viewpoint** tab ➤ **Save, Load & Playback** panel ➤ **Saved Viewpoints** tool launcher .

 **Menu:** Classic user interface: **View** ➤ **Control Bars** ➤ **Saved Viewpoints**

 **Command entry:** CTRL + F11

You get a different shortcut menu, depending on what element you right-click in the **Saved Viewpoints** window. All shortcut menus share the **Sort** option, which sorts the contents of the window alphabetically, including folders and their contents.

Blank Space

Save Viewpoint Saves the current viewpoint, and adds it to the **Saved Viewpoints** window.

New Folder Adds a folder to the **Saved Viewpoints** window.

Add Animation Adds a new empty viewpoint animation, ready for dragging viewpoints onto.

Add Cut Adds an animation cut. Cuts are used as pauses in the viewpoint animations, and are 1 second long by default.

Sort Sorts the contents of the **Saved Viewpoints** window alphabetically.

Help Opens the Help system.

Saved Viewpoint

Save Viewpoint Creates a copy of the selected viewpoint in the **Saved Viewpoints** window.

New Folder Adds a folder above the selected viewpoint.

Add Animation Adds a new empty viewpoint animation above the selected viewpoint.

Add Cut Adds an animation cut above the selected viewpoint. Cuts are used as pauses in the viewpoint animations, and are 1 second long by default.

Add Copy Creates a copy of the selected viewpoint in the **Saved Viewpoints** window. The copy is named the same as the selected viewpoint, but includes the version number in brackets. For example, View1(1), View1(2) and so on.

Add Comment Adds a comment regarding the selected viewpoint. See [Comments, Redlines, and Tags](#) (page 399) for more information on comments.

Edit Comment When available, opens the **Edit Comment** dialog box.

Edit Opens the **Edit Viewpoint** dialog box, and enables you to manually edit the viewpoint's attributes.

Update Makes the selected viewpoint the same as the current viewpoint in the **Scene View**.

Transform Opens the **Transform** dialog box. It enables you to transform the camera position. This option is not available in a 2D workspace.

Delete Deletes the selected viewpoint from the **Saved Viewpoints** window.

Rename Enables you to rename the selected viewpoint.

Copy Name Copies the name of the selected viewpoint to the Clipboard.

Sort Sorts the contents of the **Saved Viewpoints** window alphabetically.

Help Opens the Help system.

Viewpoint Animation

Save Viewpoint Saves the current viewpoint, and adds it as the last keyframe in the selected viewpoint animation.

New Folder Adds a folder as the last keyframe in the selected viewpoint animation.

Add Animation Adds a new empty viewpoint animation as the last keyframe in the selected viewpoint animation.

Add Cut Adds an animation cut to the end of viewpoint animation. Cuts are used as pauses in the viewpoint animations, and are 1 second long by default. You can drag the created cut into a different position.

Add Copy Creates a copy of the selected viewpoint animation in the **Saved Viewpoints** window. The copy is named the same as the selected viewpoint animation, but includes the version number in brackets. For example, View1(1), View1(2) and so on.

Add Comment Adds a comment regarding the selected viewpoint animation. See [Comments, Redlines, and Tags](#) (page 399) for more information on comments.

Edit Comment When available, opens the **Edit Comment** dialog box.

Edit Opens the **Edit Animation** dialog box, and enables you to set the duration of the selected viewpoint animation, the type of smoothing, and whether it loops or not.

NOTE Clicking **Edit** over an animation keyframe, opens the **Edit Viewpoint** dialog box; and clicking **Edit** over an animation cut, opens the **Edit Animation Cut** dialog box.

Update Updates all keyframes in the viewpoint animation with the current render style, lighting, and navigation tool or mode.

NOTE Clicking **Update** over a single keyframe will only update that frame with the current modes.

Transform Opens the **Transform** dialog box. It enables you to transform the camera position. This option is not available in a 2D workspace.

Delete Deletes the selected viewpoint animation from the **Saved Viewpoints** window.

NOTE Clicking **Delete** over a keyframe or a cut, removes the keyframe or cut from the viewpoint animation.

Rename Enables you to rename the selected viewpoint animation, keyframe, or cut.

Copy Name Copies the name of the selected viewpoint animation, keyframe, or cut to the Clipboard.

Sort Sorts the contents of the **Saved Viewpoints** window alphabetically.

Help Opens the Help system.

Folder

Save Viewpoint Saves the current viewpoint, and adds it to the selected folder.

New Folder Adds a subfolder to the selected folder.

Add Animation Adds a new empty viewpoint animation to the selected folder.

Add Cut Adds an animation cut to the selected folder. Cuts are used as pauses in the viewpoint animations, and are 1 second long by default. You can drag the created cut into a different position.

Add Copy Creates a copy of the selected folder in the **Saved Viewpoints** window. The copy is named the same as the selected folder, but includes the version number in brackets. For example, Folder1(1), Folder1(2) and so on.

Add Comment Adds a comment regarding the selected folder. See [Comments, Redlines, and Tags](#) (page 399) for more information on comments.

Edit Comment When available, opens the **Edit Comment** dialog box.

Update Updates all viewpoints in the folder with the current render style, lighting and navigation tool or mode. Choosing **Update** for a single viewpoint will only update that viewpoint with the current modes.

Transform Opens the **Transform** dialog box. It enables you to transform the camera position. This option is not available in a 2D workspace.

Delete Removes the selected folder and all of its contents from the **Saved Viewpoints** window.

Rename Enables you to rename the selected folder.

Copy Name Copies the name of the selected folder to the Clipboard.

Sort Sorts the contents of the **Saved Viewpoints** window alphabetically.

Help Opens the Help system.

Save Viewpoints

New viewpoints are named “ViewX” where “X” is the next available number added to the list. This new viewpoint takes all the attributes of the current viewpoint in the **Scene View**.

To save a viewpoint

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Save**

Viewpoint drop-down ► **Save Viewpoint** .

The **Saved Viewpoints** window is now in focus, and a new viewpoint is added.

- 2 Type a new name for your viewpoint in the **Saved Viewpoints** window, and press Enter.

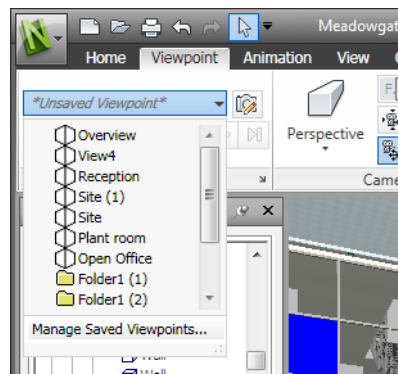
 **Menu:** Classic user interface: **Viewpoint** ► **Saved Viewpoints** ► **Save Viewpoint**

Recall Viewpoints

You can return to any of previously saved viewpoints. On recalling viewpoints the navigation mode that was active when the viewpoint was created will be re-selected. Any redlines and comments associated with the viewpoint will also be reinstated.

To recall a viewpoint from the ribbon

- Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Current Viewpoint** drop-down, and choose the saved viewpoint from the list.



It is now displayed in the **Scene View**.

To recall a viewpoint from the Saved Viewpoints window

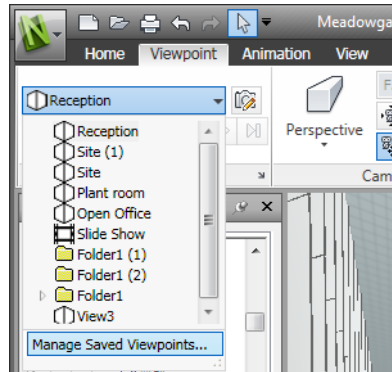
- 1 If the **Saved Viewpoints** window is not displayed, click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Saved Viewpoints** tool launcher .
- 2 Click the desired viewpoint in the list. It is now displayed in the **Scene View**.

Organize Viewpoints

Viewpoints can be organized into folders, as necessary.

To organize viewpoints into folders

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Current Viewpoint** drop-down ► **Manage Saved Viewpoints**.



This opens the **Saved Viewpoints** window, and makes it the active window.

- 2 Right-click an empty space in the **Saved Viewpoints** window, and click **New Folder**.
- 3 Type in a new name, and press Enter.

TIP Use names that can help you identify your viewpoints in future.

- 4 Drag the required viewpoints into your new folder.

Edit Viewpoints

Depending on whether you work in a 2D or 3D workspace you can edit all or some of the following viewpoints attributes, including camera position, field of view, speed of motion and saved attributes. All entries are measured in [Display Units](#) (page 157).

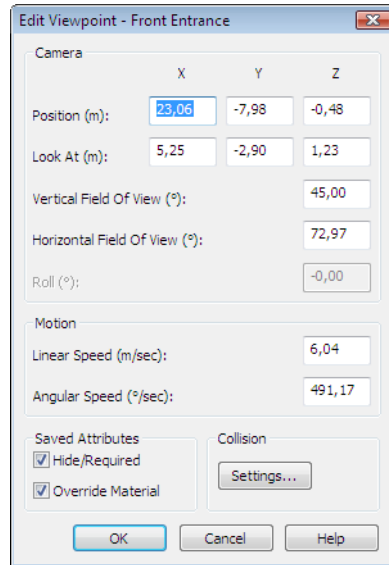
TIP Click **Viewpoint** tab and slide out of the **Navigate** panel to quickly adjust linear and angular speed of motion for your current viewpoint in a 3D workspace.

To edit current viewpoint

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Edit**

Current Viewpoint

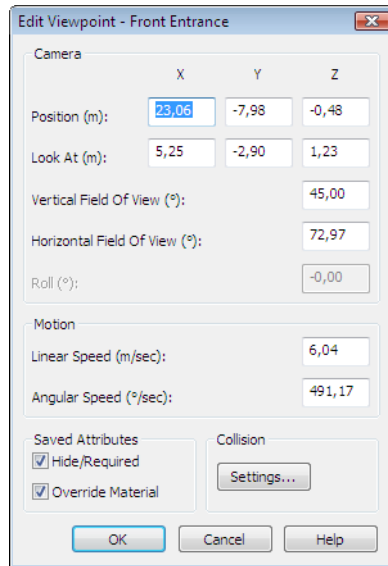
- 2 Use the [Edit Viewpoint dialog box](#) (page 751) to adjust the viewpoint's attributes.



- 3 Click **OK**.

To edit a viewpoint

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Current Viewpoint** drop-down ► **Manage Saved Viewpoints**.
- 2 In the **Saved Viewpoints** window, right-click the viewpoint you want to modify, and click **Edit**.
- 3 Use the [Edit Viewpoint dialog box](#) (page 751) to adjust the viewpoint's attributes.



4 Click **OK**.

To delete a viewpoint

- 1 Click **Viewpoint** tab ► **Save, Load & Playback** panel ► **Current Viewpoint** drop-down ► **Manage Saved Viewpoints**.
- 2 In the **Saved Viewpoints** window, right-click the viewpoint you want to remove, and click **Delete**.

Default Viewpoint Options

Two view attributes can be saved with a viewpoint:

- **Hide/Required** - whether items are hidden or required.
- **Override Materials** - the color and transparency of items.

You can set a viewpoint to save either attribute by [editing](#) (page 449) the viewpoint. To update changes to overridden material or hide/required, use the **Update** option on the viewpoints shortcut menu. Be careful, though, as this also updates the point of view as well, which may disrupt any redline information stored with the viewpoint.

By default these attributes are not stored with new saved viewpoints. If you do wish them to be saved by default, then this can be set in the **Options Editor**.

Default collision settings can also be saved with a viewpoint, including whether collision, gravity, crouching, and third person view are enabled. These settings are only used in a 3D workspace.

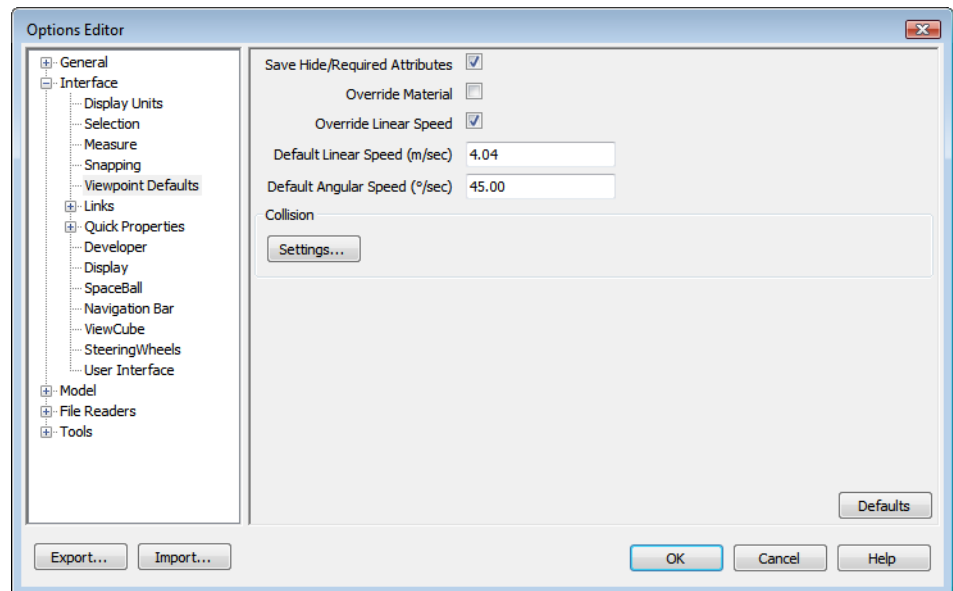
You can set a viewpoint to save any of these settings by editing the viewpoint, in the same way as the view attributes.

By default, all of the collision settings are disabled. If you want to save your preferred collision setting defaults, use the **Options Editor**.

NOTE When you modify the default viewpoint settings, your changes do not affect the currently opened Autodesk Navisworks file. They are used as soon as you open a new Autodesk Navisworks file, or start a new Autodesk Navisworks session.

To set the default view attributes

- 1 Click the application button  ➤ **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Viewpoint Defaults** option.



- 3 Select the **Save Hide/Required Attributes** check box if you want to save the hidden and required items with the viewpoints you save. This means that when returning to those viewpoints, the items that were hidden when the viewpoint was saved will be hidden again and those that were drawn will be drawn again. By default, this check box is clear, as it requires a relatively large amount of memory to save this state information with each viewpoint.

See [Hide Objects](#) (page 348) for more information on hidden items, and [Make Objects Required](#) (page 330) for more information on making items required.

- 4 Select the **Override Material** check box if you want to save the material overrides with the viewpoints you save. This means that when returning to those viewpoints, the material overrides set when the viewpoint was saved will be reinstated. By default, this check box is clear as it requires a relatively large amount of memory to save this state information with each viewpoint.

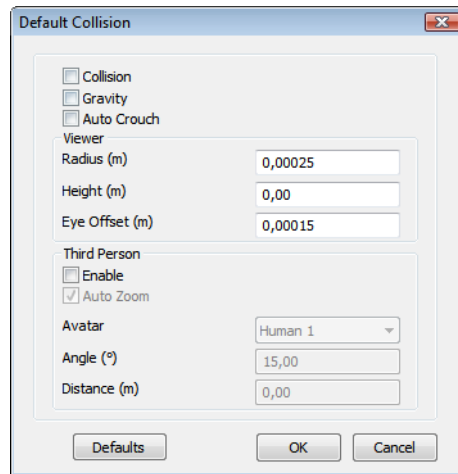
See [Manipulate Object Attributes](#) (page 382) for more information on overriding materials.

- 5 Select the **Override Model Linear Speed** check box to be able to set a specific speed to navigate on loading a model. Without this selected, the linear navigation speed is directly related to the size of the model loaded.

- 6 The **Default Angular Speed** can be set to any number of degrees per second. This affects the speed at which the camera turns.
- 7 Click **OK**.

To set the default collision options

- 1 Click the application button  > **Options**.
- 2 In the **Options Editor**, expand the **Interface** node, and click the **Viewpoint Defaults** option.
- 3 On the **Viewpoint Defaults** page, click the **Settings** button.
- 4 In the [Default Collision dialog box](#) (page 747), select the default options you want Autodesk Navisworks to initialize with.



- 5 Click **OK** to return to the **Options Editor**.
- 6 Click **OK** to save the changes.

Share Viewpoints

You can export saved viewpoints from Autodesk Navisworks into an XML file, and share them with other users.

For more information, see [Export Viewpoints](#) (page 498) and [Import Viewpoints](#) (page 488).

Sectioning

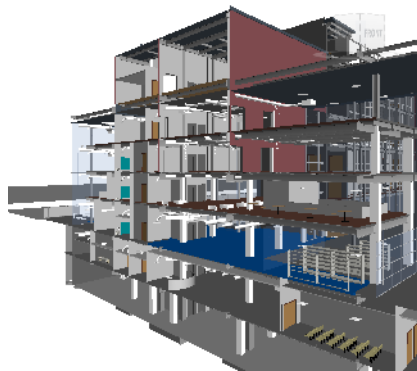
Autodesk Navisworks enables you to turn on sectioning for the current viewpoint and to create cross sections of your model in a 3D workspace. The sectioning functionality is not available for 2D sheets.

A cross section is a cut-away view of a 3D object that enable you to see inside it. You can turn sectioning on and off for the current viewpoint by clicking

Viewpoint tab ► **Sectioning** panel ► **Enable Sectioning** . When the sectioning is turned on, the Sectioning Tools contextual tab is automatically displayed on the ribbon.

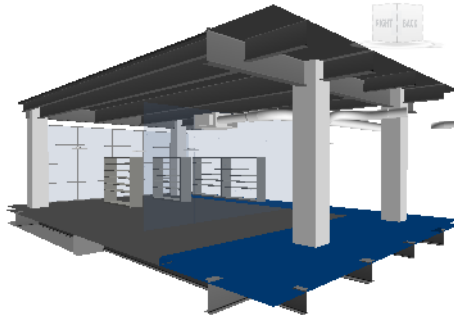
There are two sectioning modes available from the **Sectioning Tools** tab ► **Mode** panel: **Planes** and **Box**.

Planes  mode allows you to make up to six sectional cuts in any plane while still being able to navigate around the scene, enabling you to see inside models without hiding any item. By default section planes are created through the center of the visible area of the model.



Section planes are stored inside viewpoints and so can also be used within viewpoint animations and object animations to show a dynamically sectioned model. See [Saved Viewpoints Window](#) (page 442) for more information on viewpoints, and [Record and Play Animations](#) (page 471) for more information on animations.

Box  mode enables you to focus your review on specific and limited areas of the model. As you move the box around, only the geometry within the defined section box is displayed in the **Scene View**.



To turn sectioning on or off for the current viewpoint

■ Click **Viewpoint** tab ► **Sectioning** panel ► **Enable Sectioning** 

 **Toolbar:** Classic user interface: **Workspace** ► **Enable Sectioning** 

Enable and Use Section Planes

To view cross-sectional cuts of your model, you can enable up to six section planes. When a plane is ‘enabled’ it means that it affects (cuts through) the scene. The current plane is the one that is rendered visibly in the **Scene View**. Selecting a plane as current automatically enables that plane if it was not already enabled.

When a section plane is first enabled, it is created with the default alignment and position. Afterwards, enabling a section plane restores the saved alignment, position, and rotation if it is available for the current viewpoint.

NOTE To save the settings for enabled section planes (alignment, position, rotation), you need to save the current viewpoint with sectioning enabled and in sectioning mode.

By default, a section plane is created within the view, and as close to the center of the view as possible. Visually, a section plane is represented by a light blue wireframe. You can hide the visual plane representation by toggling the corresponding gizmo button.

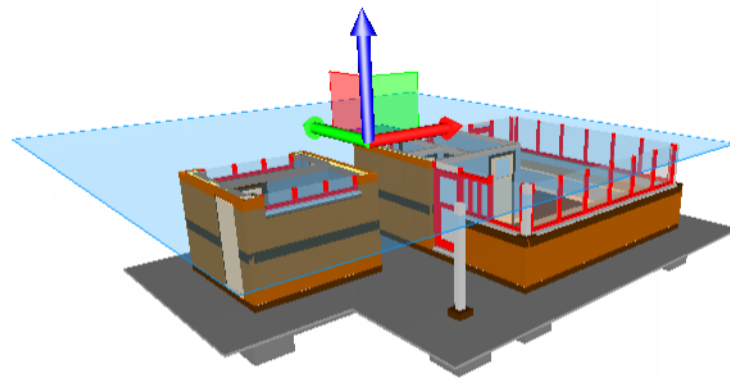
To use a plane to cross-section a 3D model for the first time

- 1 Click **Viewpoint** tab ► **Sectioning** panel ► **Enable Sectioning**



Autodesk Navisworks opens the **Sectioning Tools** tab on the ribbon, and draws a section plane through the model in the **Scene View**. Default alignment for Plane 1 is Top. Default position is through the center of the visible area of the model. Move is the default gizmo.

- 2 Drag the gizmo to position the current plane as needed.

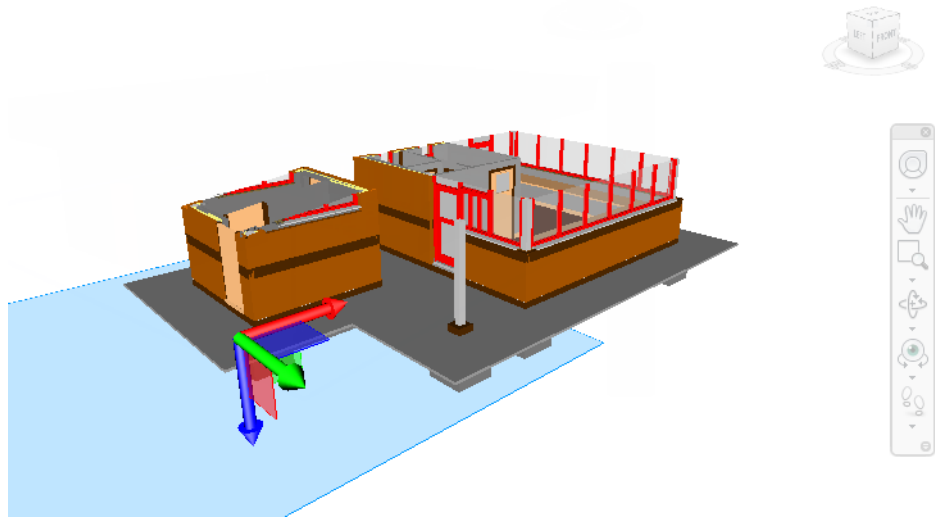


- 3 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save**

Viewpoint  to save the current sectioned viewpoint.

To change the current section plane

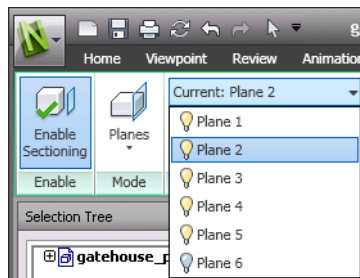
- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to make current, for example, Plane 2.
The selected plane is now visible (light blue wireframe), and can be manipulated. The other section plane is not visible but still cuts through the model in the **Scene View** for as long as the plane is enabled.



TIP To see which planes are enabled, expand the **Current Plane** drop-down on the **Planes Settings** panel. Enabled plane has a lit light bulb  icon next to its name.

To enable or disable additional section planes

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
 - 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and click the bulb icons next to all required planes.
- When a bulb is lit, the corresponding section plane is enabled and cuts through the model in the **Scene View**.



NOTE If you want to move all planes together, you can link them. Otherwise, you can only move one (the current) plane at a time.

Customize Section Plane Alignment

By default, section planes are mapped to one of the six primary directions as follows:

Plane No	Plane Name	Default Alignment
1	Plane 1	Top
2	Plane 2	Bottom
3	Plane 3	Front
4	Plane 4	Back
5	Plane 5	Left
6	Plane 6	Right

You can select a different alignment for the current section plane. There are 6 fixed alignments and 3 custom alignments to choose from:

- **Top**  - aligns the current plane to the top of model.
- **Bottom**  - aligns the current plane to the bottom of model.
- **Front**  - aligns the current plane to the front of model.
- **Back**  - aligns the current plane to the back of model.
- **Left**  - aligns the current plane to the left of model.
- **Right**  - aligns the current plane to the right of model.
- **Align To View**  - aligns the current plane to the current viewpoint camera.
- **Align To Surface**  - enables you to pick a surface, and place the current plane 'on' that surface, with its normal aligned to the normal of the triangle picked.

- **Align To Line**  - enables you to pick a line, and place the current plane 'on' that line, at the point where you clicked, and aligned so that its normal is on the line itself, facing towards the camera.

To align a section plane to one of the pre-fixed directions

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to customize, for example, Plane 1.
This plane becomes the current plane.
- 3 Click the **Alignment** drop-down on the **Planes Settings** panel, and select one of the six pre-fixed directions, for example, Top 
Autodesk Navisworks updates the position and alignment of the section plane.
- 4 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To align a section plane to line

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to customize, for example, Plane 1.
This plane becomes the current plane.
- 3 Click the **Alignment** drop-down on the **Planes Settings** panel, and click **Align To Line** 
The cursor changes to a target .
- 4 In the **Scene View**, click somewhere on the line you want to align to.
Autodesk Navisworks updates the position and alignment of the section plane, so that it is placed at the point you clicked.
- 5 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To align a section plane to surface

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .

- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to customize, for example, Plane 1. This plane becomes the current plane.
- 3 Click the **Alignment** drop-down on the **Planes Settings** panel, and click **Align To Surface** .

The cursor changes to a crosshair .

- 4 In the **Scene View**, click somewhere on an object you want to align to. Autodesk Navisworks updates the position and alignment of the section plane so that it is placed 'on' the surface of the selected object.
- 5 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To align a section plane to view

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to customize, for example, Plane 1. This plane becomes the current plane.
- 3 Click the **Alignment** drop-down on the **Planes Settings** panel, and click **Align to View** .
- 4 Optional: Click **Sectioning Tools** ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

Move and Rotate Section Planes

The section planes can be manipulated with the sectioning gizmos, available from the **Sectioning Tools** tab ► **Transform** panel. You can also manipulate the section box numerically. Section planes can be moved and rotated but cannot be scaled.

By default, when a section plane is made current, the move gizmo is used, unless you have already selected the rotate gizmo before activating the section plane. All gizmos share the same location/rotation. This means that moving one gizmo affects the position of others.

You can only manipulate one plane at a time (the current plane), but it is possible to link section planes together to form slices.

The following table illustrates how the sectioning gizmos work with the section planes.

Gizmo	Behavior
Move	Perpendicular arms and faces move the current plane and the gizmo. Parallel arms and face move only the gizmo.
Rotate	The position of the gizmo signifies the center of rotation. Perpendicular arcs rotate the current plane and the gizmo. Parallel arc rotates only the gizmo. Arms move only the gizmo.
Scale	Is not used with section planes.

To move a section plane with gizmo

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
 - 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to use, for example, Plane 3.
This plane becomes the current plane.
 - 3 If the move gizmo is not visible in the **Scene View**, click **Transform** panel ► Move .
 - 4 Drag the gizmo to move the current plane as needed.
- TIP** Perpendicular arms and faces move the current plane and the gizmo. Parallel arms and face move only the gizmo.

To move a section plane numerically

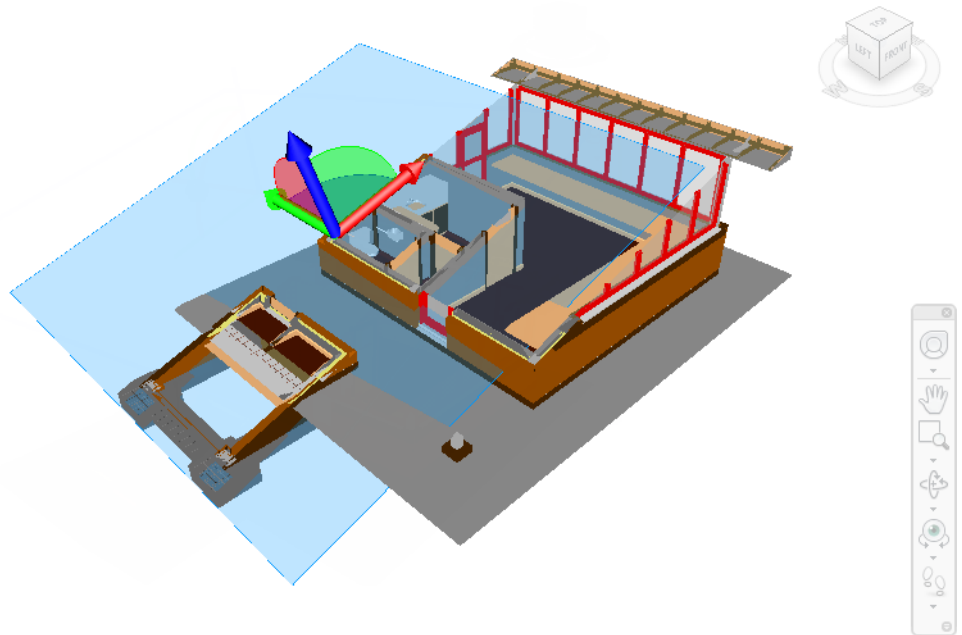
- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to use, for example, Plane 3.
This plane becomes the current plane.

- 3 Slide out the **Transform** panel, and type in numerical values into the **Position** manual entry boxes to move the current plane by the amount entered.

To rotate a section plane with gizmo

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to use, for example, Plane 3.
This plane becomes the current plane.
- 3 Click **Sectioning Tools** tab ► Click **Transform** panel ► **Rotate** .
- 4 Drag the gizmo to rotate the current plane as needed.

TIP Perpendicular arcs rotate the current plane and the gizmo. Parallel arc rotates only the gizmo. Arms move only the gizmo.



To rotate a section plane numerically

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Click the **Current Plane** drop-down on the **Planes Settings** panel, and select the plane you want to use, for example, Plane 3.
This plane becomes the current plane.
- 3 Slide out the **Transform** panel, and type in numerical values into the **Rotation** manual entry boxes to rotate the current plane by the amount entered.

Link Section Planes

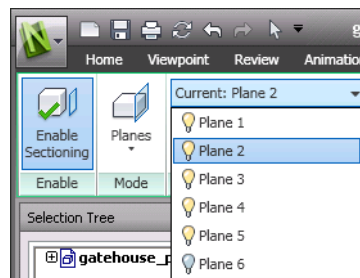
In Autodesk Navisworks, you can enable up to 6 planes to cut through your model but only the current plane can be manipulated with the sectioning gizmos.

Linking section planes together make them move as one, and enables you to quickly slice your model in real time. The slices can be used in viewpoints, viewpoint animations, and object animations.

To link planes together

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Planes** .
- 2 Enable the required planes by click the **Current Plane** drop-down on the **Planes Settings** panel, and click the bulb icons next to all required planes.

When a bulb is lit, the corresponding section plane is enabled and cuts through the model in the **Scene View**.



- 3 Click **Link Section Planes** on the **Planes Settings** panel. All enabled planes are linked now together into a slice.
- 4 If the move gizmo is not visible in the **Scene View**, on the **Transform** panel, click **Move**  .
- 5 Drag the gizmo to move the current section plane. All section planes are now moving together, effectively creating a slice through the model.



- 6 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

NOTE You can click **Animation** tab ► **Create** panel ► **Record**  , and create a viewpoint animation showing the model as it is sliced.

Enable and Use Section Box

You can use a section box to focus your review on specific and limited areas of the model. The section box can be moved, rotated, and scaled with the sectioning gizmos, available from the **Sectioning Tools** tab ► **Transform** panel. You can also manipulate the section box numerically.

By default, when a section box is enabled, the move gizmo is used, unless you have already selected a different gizmo before activating the section box. All

gizmos share the same location/rotation. This means that moving one gizmo affects the position of others.

When the section box is first created, the default size of the box is based upon the bounds of the current viewpoint. The box is created to fill the view such that no part of the box is drawn off-screen. Afterwards, enabling a section box restores the saved position, rotation, and scale information used if it is available for the current viewpoint.

NOTE To save section box settings (position, rotation, size), you need to save the current viewpoint with sectioning enabled and in box mode.

The following table illustrates how the sectioning gizmos work with the section box.

Gizmo	Behavior
Move	You can use arms and faces to move both the box and the gizmo itself in the relevant axis.
Rotate	The position of the gizmo signifies the center of rotation. Arcs rotate the box. Arms move the gizmo only.
Scale	You cannot move the transform center. Pulling on any of the scale points scales the box.

To use a box to cross-section a 3D model for the first time

- 1 Click **Viewpoint** tab ► **Sectioning** panel ► **Enable Sectioning**

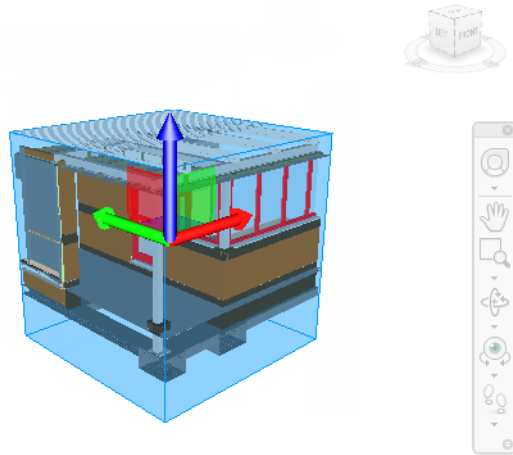


Autodesk Navisworks opens the **Sectioning Tools** tab on the ribbon, and draws a section plane through the model in the **Scene View**.

- 2 Click **Sectioning Tools** tab ► **Mode** panel ► **Box**

The box is now visually present on the screen, with the move gizmo enabled by default.

- 3 Drag the gizmo to box-section your model along the axes.



- 4 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To move the section box with gizmo

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 On the **Transform** panel, click **Move** .
- 3 Drag the gizmo arms or faces to move the box as needed.

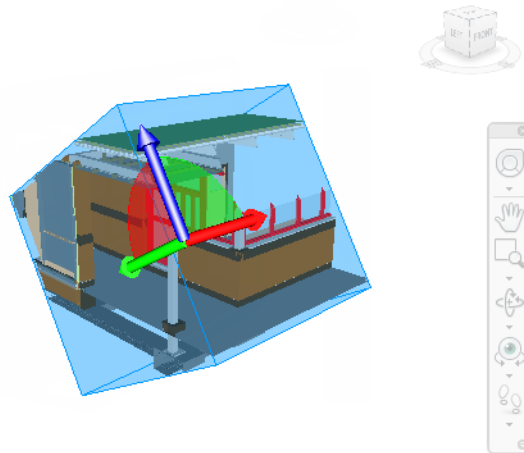
To move the section box numerically

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 Slide out the **Transform** panel, and type in numerical values into the **Position** manual entry boxes to move the box by the amount entered.

To rotate the section box with gizmo

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 On the **Transform** panel, click **Rotate** .

- 3 Drag the gizmo to rotate the box as needed.



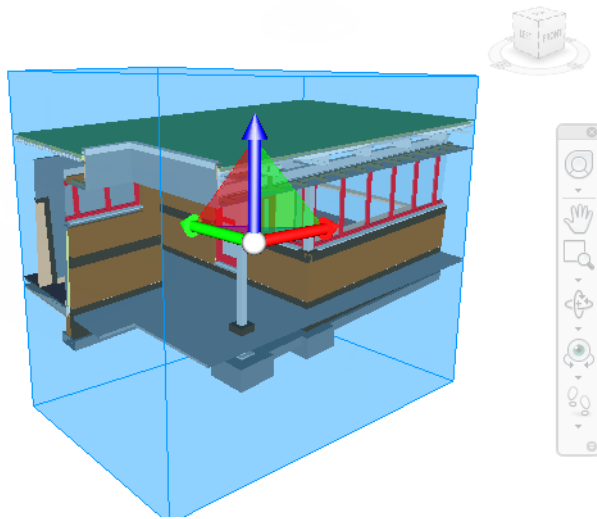
- 4 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To rotate the section box numerically

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 Slide out the **Transform** panel, and type in numerical values into the **Rotation** manual entry boxes to rotate the box by the amount entered.

To scale the section box with gizmo

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 On the **Transform** panel, click **Scale** .
- 3 Pull the scale points on the gizmo to resize the box as needed.



- 4 Optional: Click **Sectioning Tools** tab ► **Save** panel ► **Save Viewpoint**  to save the current sectioned viewpoint.

To scale the section box numerically

- 1 Click **Sectioning Tools** tab ► **Mode** panel ► **Box** .
- 2 Slide out the **Transform** panel, and type in numerical values into the **Size** manual entry boxes to scale the box by the amount entered.

Record and Play Animations

9

In Autodesk Navisworks there are two types of animation: viewpoint animation and object animation.

Viewpoint Animation

Viewpoint animation is a quick and efficient way of recording both your movement through the model and views of the model. There are two ways to produce a viewpoint animation in Autodesk Navisworks:

- By recording interactive navigation
- By creating animated transitions between saved views

You can also create slide show animations, which are, essentially, viewpoint transitions with a number of cuts (pauses) added between viewpoints.

Object Animation

In addition to viewpoint animation, you can also animate and interact with 3D geometry in your model. Although the ability to record animations and produce scripts necessary to interact with animated objects is limited to users with access to the **Animator** and **Scripter** functionality, any user can actually play them back once they have been recorded.

See [Animate Objects](#) (page 501) for step-by-step instructions on how to record animations and scripts.

Create and Edit Viewpoint Animations

There are two ways to create viewpoint animations in Autodesk Navisworks. You can either simply record your real-time walk through, or you can assemble

specific viewpoints for Autodesk Navisworks to interpolate into a viewpoint animation later.

Viewpoint animation is controlled through the **Animation** tab and the **Saved Viewpoints** window.

It is worth remembering that you can hide items in viewpoints, override colors and transparencies and set multiple section planes and these will all be respected by a viewpoint animation. This way you can easily create powerful viewpoint animations.

Once a viewpoint animation is recorded, you can edit it to set the duration, the type of smoothing and whether it loops or not.

There is also nothing to stop you from copying viewpoint animations (hold down the CTRL key when dragging an animation on the **Saved Viewpoints** window), dragging frames off the animation into a blank space on the **Saved Viewpoints** window to remove them from the viewpoint animation, editing individual frames attributes, inserting cuts or dragging other viewpoints or viewpoint animations onto the existing one, to continue developing your animations.

Animation Cuts (Pauses)

Cuts in a viewpoint animation are simply points where the camera pauses for a while. They are inserted automatically when you click **Pause** during the interactive recording of a viewpoint animation, or you can insert them manually into an existing viewpoint animation.

To create a viewpoint animation in real time

- 1 Click **Animation** tab ► **Create** panel ► **Record**  .
To the far right of the **Animation** tab, notice that the **Recording** panel displays.
- 2 Navigate around in the **Scene View** while Autodesk Navisworks records your movement. You can even move the section planes through the model during your navigation, and this will be recorded into the viewpoint animation too.
- 3 At any point during the navigation, click **Animation** tab ► **Recording** panel ► **Pause**  .
This will pause the recording while you maneuver into a new position. To continue recording the viewpoint animation, click **Pause** again.

The resulting viewpoint animation will contain a [cut](#) (page 471) for the duration of the pause.

- 4 When finished, click **Animation** tab ► **Recording** panel ► **Stop**



The animation is saved automatically in the **Saved Viewpoints** window (click **View** tab ► **Workspace** panel ► **Windows** drop-down ► **Saved Viewpoints**). Your new viewpoint animation is called “AnimationX”, where “X” is the latest available number. The name will be editable at this point if you want to name it yourself. This viewpoint animation will also become the current active animation in the **Available Animations** drop-down list on the **Playback** panel of the **Animation** tab.

While the above method is useful for creating quick viewpoint animations on the fly, sometimes you need more control over the viewpoint camera. To do this in Autodesk Navisworks, you need to set up several viewpoints and add them to an empty viewpoint animation. When playing back the animation, Autodesk Navisworks will then interpolate between these viewpoints.

To create an animation frame by frame

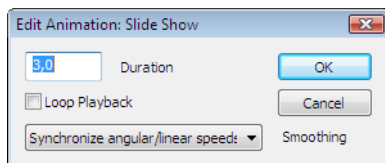
- 1 If necessary, display the **Saved Viewpoints** window (click **View** tab ► **Workspace** panel ► **Windows** drop-down ► **Saved Viewpoints**).
- 2 Right-click the **Saved Viewpoints** window and select **Add Animation**.
A new viewpoint animation is created, called “AnimationX”, where “X” is the latest available number. The name will be editable at this point if you want to name it yourself. There will be no plus sign next to the new viewpoint animation because it is empty.
- 3 Navigate to a position in the model you would like to add to your animation and save the new location as a viewpoint. (Right-click the **Saved Viewpoints** window and select **Save Viewpoint**.) Repeat this step as desired. Each viewpoint will become a frame for the animation. The more frames you have, the smoother and more predictable the viewpoint animation will be. See [Save Viewpoints](#) (page 447) for more information on creating viewpoints.
- 4 When you have all required viewpoints, drag them onto the empty viewpoint animation you just created. You can drag them on one-by-one, or you can select multiple viewpoints using the CTRL and SHIFT keyboard keys and drag several on at once.

If you drop them onto the viewpoint animation icon itself, then the viewpoints will become frames at the end of the animation, but you can drop the viewpoints anywhere on the expanded animation to put them where you wish.

- 5 At this point, you can use the **Playback Position** slider on the **Playback** panel of the **Animation** tab to move backward and forward through the viewpoint animation to see how it looks.
- 6 You can edit any of the viewpoints inside the viewpoint animation (see [Edit Viewpoints](#) (page 449) for details on this), or you can add more viewpoints, delete them, move them around, add cuts, and [edit the animation itself](#) (page 474) until you are happy with the viewpoint animation.
- 7 Once you have several viewpoint animations, you can drag and drop them onto a master viewpoint animation to compose more complex combinations of animations, just like dragging and dropping viewpoints onto an animation as a frame.

To edit a viewpoint animation

- 1 If necessary, display the **Saved Viewpoints** window (click **View** tab ► **Workspace** panel ► **Windows** drop-down ► **Saved Viewpoints**).
- 2 In the **Saved Viewpoints** window, right-click the viewpoint animation you want to modify and select **Edit**.
- 3 In the **Edit Animation** dialog box, in the **Duration** text box, type the desired duration in seconds.



- 4 If you want the viewpoint animation to play continuously, select the **Loop Playback** check box.
- 5 From the **Smoothing** drop-down list, select the type of smoothing you want the viewpoint animation to use.

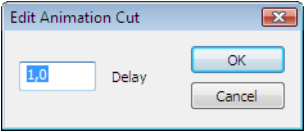
None means that the camera will move from one frame to the next without any attempt at smoothing out the corners.

Synchronize Angular/Linear Speeds will smooth the differences between the speeds of each frame in the animation, resulting in a less jerky animation.

6 Click **OK**.

To insert cuts (pauses) into a viewpoint animation

- 1 If necessary, display the **Saved Viewpoints** window (click **View** tab ► **Workspace** panel ► **Windows** drop-down ► **Saved Viewpoints**).
- 2 Below the animation frame where you want to insert the cut, right-click and select **Add Cut**.
- 3 Type in the name of the cut, or press **Enter** to accept the default name, which will be “CutX”, where “X” is the next available number.
- 4 The default duration of a cut is 1 second. To alter the duration of this pause, right-click the cut and select **Edit**.
- 5 In the **Edit Animation Cut** dialog box, in the **Delay** text box, type the desired duration of the pause in seconds.



6 Click **OK**.

Play Animations and Scripts

You can play back both pre-recorded object animation and viewpoint animation in the **Scene View**.

The viewpoint animations play in real time; this means that the Autodesk Navisworks engine is still attempting to maintain the guaranteed frame rate so some drop-out may still occur, just as in real-time navigation.

Control	Purpose
	Rewinds the current animation back to the beginning.
	Steps back a single animation frame or keyframe.
	Plays the current animation backwards.

Control	Purpose
	Records the viewpoint animation.
	Stops animation playback or recording.
	Pauses animation playback or recording.
	Plays the currently selected animation.
	Steps forwards one frame or keyframe in the animation.
	Moves forward to the end of animation.
	The playback time slider.
	The playback time spinner.

To play an animation

- 1 Click **Animation** tab ► **Playback** panel ► **Available Animations** drop-down list and select the animation you want to play back.
- 2 On the **Playback** panel, click **Play** ► .
 Use the VCR buttons on the **Playback** panel to control the animation. The **Playback Position** slider enables you to quickly move forward and backward through the animation. Full left is at the beginning and full right is at the end.
 To the right of the **Playback Position** slider, there are two animation progress indicators: percentage and time (in seconds). You can type a number into either box to set the camera at a certain point.
- 3 For viewpoint animations, you may notice that the frame in the animation in the **Saved Viewpoints** window (click **View** tab ► **Workspace** panel ► **Windows** drop-down ► **Saved Viewpoints**) is highlighted when the animation is playing. Click any frame to set the

camera to that point in time in the viewpoint animation and continue playing back from there.

To enable animation scripts

- Click **Animation** tab ► **Scripts** panel ► **Enable Scripts**  .

You can now interact with your model. For example, if there is a script to open a door on pressing a specific key on the keyboard, pressing this key will open the door.

Share Animations

You can export the animations to AVI files for playback with Windows Media Player.

For more information, see [Export Images and Animations](#) (page 494).

Work Within a Team

10

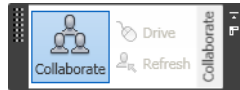
Autodesk Navisworks Manage 2012 enables multiple users to participate in a single design review session across a Local Area Network (LAN).

The Collaborate tool has two noteworthy limitations.

- Collaboration between different versions of Autodesk Navisworks is not supported.
- This feature is only available for Windows XP users, as it utilizes the shared program features of Windows NetMeeting, which is unavailable in the Vista and Windows 7 operating systems.

Collaborate Panel

The collaboration tools necessary to run collaboration sessions are located on the **Review** tab ► **Collaborate** panel.



By default, the **Collaborate** panel is not displayed. To display it, right-click the **Review** tab, and click **Show Panels** ► **Collaborate** on the shortcut menu.

Collaboration Session

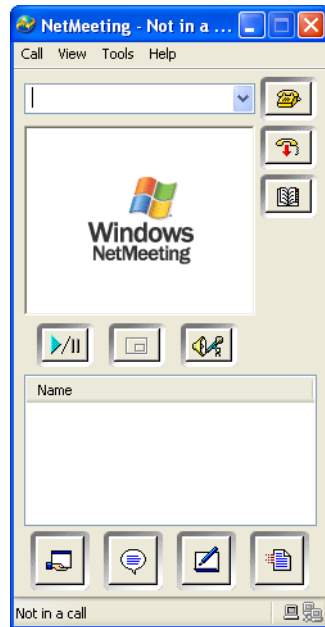
All meeting participants require access to a Autodesk Navisworks NWF or NWD file, in a shared location. One of the participants will host the meeting and place a call to invite the others to join the meeting. Any of the participants who have joined the meeting can take control and drive the session. All navigation performed by the driver will be displayed in the **Scene View** on each of the

participants' machines. Any viewpoints or redlines, for example, added during the session can be updated on all participants' machines at the click of a button.

NOTE If a collaborative review session, as outlined here, is not conducted in single room, then additional teleconferencing provisions could be necessary. This may be using the NetMeeting Whiteboard, or your own telephone system.

To start a collaboration session

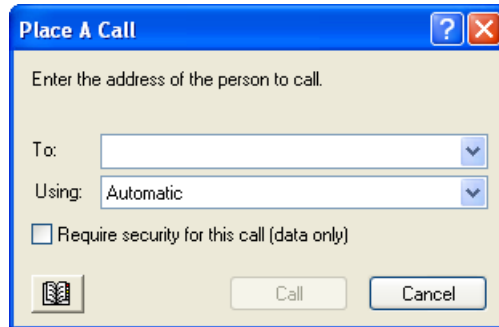
- 1 Open the Autodesk Navisworks file that you want to collaborate on from a shared directory.
- 2 Click **Review** tab ► **Collaborate** panel ► **Collaborate**  .
This initializes Windows® NetMeeting®.



NOTE The first time Windows NetMeeting initializes, a Setup wizard will take you through the setup process. You will need to enter your name and email address. When using NetMeeting on a LAN you do not need to log onto a directory server, as these will not be available to you.

To place a call, inviting attendees to join

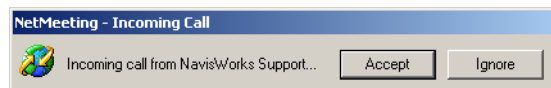
- 1 Having initialized Windows NetMeeting, click the **Call** button  in the **NetMeeting** dialog box.
- 2 In the **Place a Call** dialog box, enter the machine name or IP address of the machine you wish to join the meeting into the **To** box.



- 3 Click the **Call** button to send the invite.
Once the person receiving the invite accepts this, both their name and yours will be listed in the NetMeeting dialog box.
- 4 Repeat this procedure to invite all required participants.

To accept an invitation

- 1 When you are invited to join a meeting, the **Incoming Call** dialog box is displayed.



- 2 Click the **Accept** button to join the meeting, or **Ignore** to decline the invitation.

NOTE Once you have accepted a call, you will need to click **Review** tab

➤ **Collaborate** panel ➤ **Collaborate**  to start your own collaboration session.

To become the driver

During a collaboration meeting, anyone in the call can take control of the session and become the driver. The driver will control navigation of the shared model on all machines in the call.

- 1 Click **Review** tab ➤ **Collaborate** panel ➤ **Drive** .
- 2 Upon clicking the drive button, all other users in the call will receive a message advising that you are requesting control. They will have to answer **Yes** to this message if you are to drive Autodesk Navisworks on their machine.

To refresh all attendees machines

Although real-time navigation in Autodesk Navisworks can be performed on all machines in a call by one user, it is not possible for review data such as saved viewpoints, comments and redlines, to be automatically updated on all users' machines. This information can, however, be updated on their machines by refreshing the model. This refresh process can be performed on one users machine and refresh all machines in the call.

- Click **Review** tab ➤ **Collaborate** panel ➤ **Refresh** .

Share Data



Print

You can print a hard copy of the current viewpoint to any printer or plotter.

Print Preview

Before you print out a copy of the model or sheet you are working on, you may wish to see how it will appear.

To preview model/sheet before printing

- 1 Click **Output** tab ➤ **Print** panel ➤ **Print Preview** .
- 2 Use the **Zoom In** and **Zoom Out** buttons to do just that with the preview image.
- 3 Click **Print**.
- 4 In the **Print** dialog box, click **OK**.

Print Setup

This option enables you to the set up paper size and orientation options.

To change the print setup

- 1 Click **Output** tab ➤ **Print** panel ➤ **Print Settings** .

- 2 In the **Print Setup** dialog box, make changes as required to the paper and orientation.
- 3 Click the **Properties** button if you want to change printer-specific settings.
- 4 Return to the **Print** dialog box, and click **OK**.

Print Current Viewpoint

When the print option is selected, Autodesk Navisworks prints the current viewpoint scaled to fit and centered on the page.

NOTE If you would prefer to export an image for printing, see [Export an Image](#) (page 494) for more information.

To print the current viewpoint

- 1 Click **Output** tab ► **Print** panel ► **Print** .
- 2 Check the printer settings are as required, and click **OK**.

NOTE The maximum image size is 2048x2048 pixels.

The **Properties** button controls printer-specific ink and paper settings.

 **Toolbar:** Classic user interface: **Standard** ► **Print** 

Import Files

The import option inputs Intergraph PDS review data, including: PDS Tags (.tag) and PDS Display Sets (.dst).

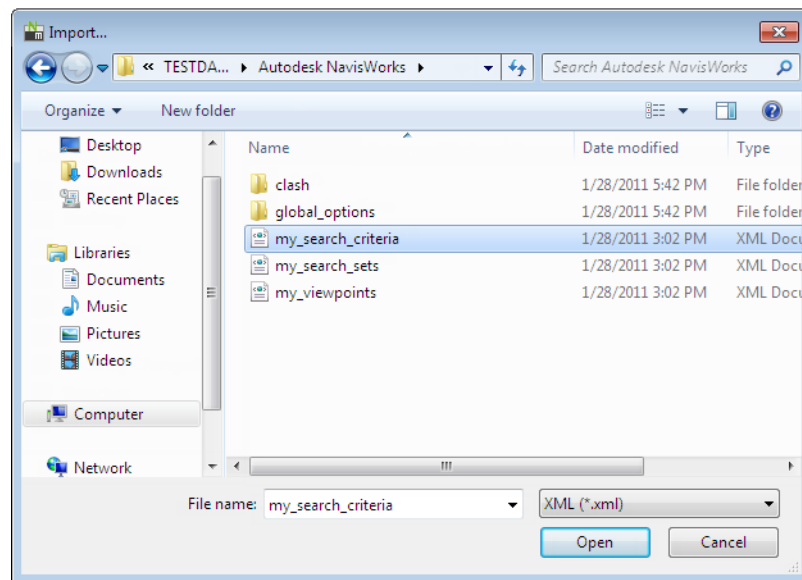
It is also possible to import various data that has been exported from previous Autodesk Navisworks sessions. You can also import 2D sheets and 3D models from DWF, DWF(x), NWD, or NWF files to the file currently open in Autodesk Navisworks. See [Add Sheets/Models to the Currently Opened File](#) (page 214).

Search Criteria Files

Search criteria can be imported into Autodesk Navisworks. This will populate the **Find Items** window. The search can then be run on the current model, finding any items that match the specific criteria. See [Find Items Window](#) (page 349) for more information on searching the model for items based on their properties.

To import saved search criteria

- 1 Click the application button  **> Import > Search** .
- 2 In the **Import** dialog box, locate the desired search XML file.



- 3 Click **Open**.

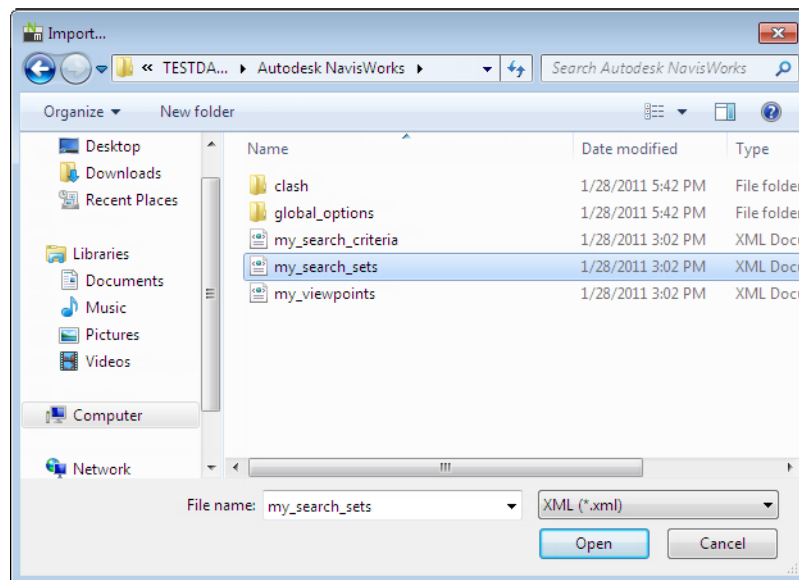
Search Set Files

Search sets can be imported into Autodesk Navisworks. This will populate the **Sets** window with pre-defined search sets (see [Sets Window](#) (page 360)). Selecting an imported **Search Set** will define the current **Find Items** criteria

and search the current model accordingly. See [Find Items Window](#) (page 349) for more information on searching the model for items based on their properties.

To import saved search sets

- 1 Click the application button  **> Import > Search Sets** .
- 2 In the **Import** dialog box, locate the desired search sets XML file.



- 3 Click **Open**.

PDS Display Set Files

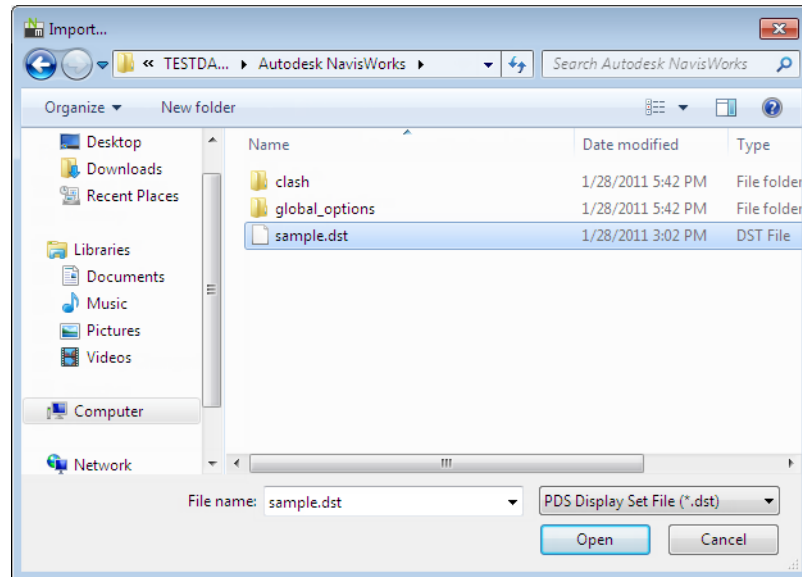
Display sets from Intergraph PDS contain detailed criteria, defining item selections. When imported into Autodesk Navisworks, DST files create **Search Sets** in the **Selection Sets** window.

See also:

[Sets Window](#) (page 360)

To import PDS display sets

- 1 Click the application button  ► **Import** ► **PDS Display Sets** .
- 2 Use the **Import** dialog box to locate the desired DST file.



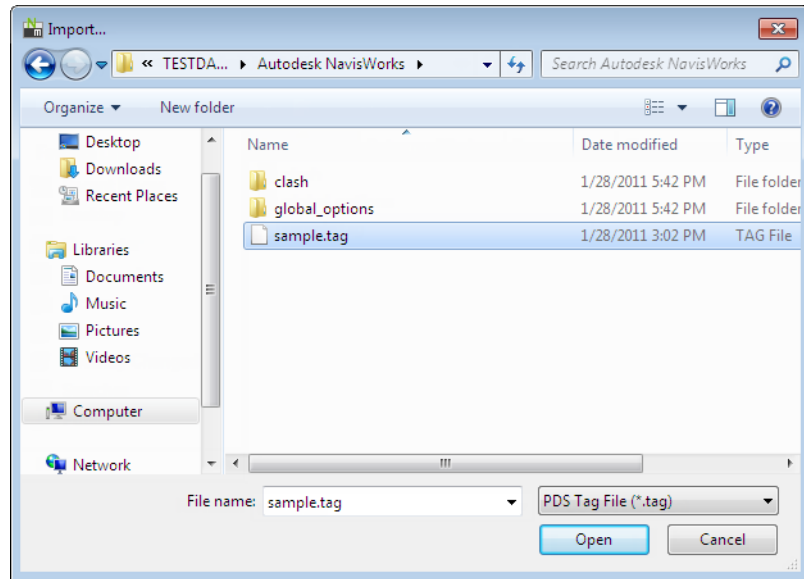
- 3 Click **Open** to import the PDS display sets.

PDS Tag Files

Tag information from Intergraph PDS contains a unique ID, saved viewpoint and corresponding comments. Tag information created in Autodesk Navisworks may also be exported to be used in Intergraph PDS. See [PDS Tag Files](#) (page 500).

To import a PDS tag file

- 1 Click the application button  ► **Import** ► **PDS Tags** .
- 2 Use the **Import** dialog box to locate the desired TAG file.



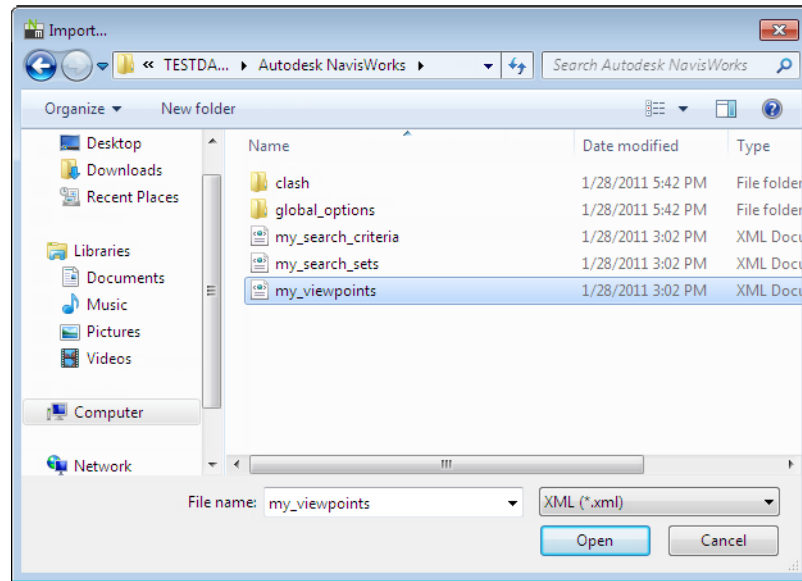
- 3 Click **Open** to import the PDS data.

Viewpoints Files

Viewpoints can be imported into Autodesk Navisworks via an XML file, enabling you to bring viewpoints into the current scene from another model file. For example, if you are working on different versions of the same model, you can save viewpoints in one version of the file, export them and then import them into the other version. See [Save Viewpoints](#) (page 447) for more information on saving viewpoints, and [Export Viewpoints Files](#) (page 498) for more information on exporting viewpoints to an XML file.

To import viewpoints and associated data

- 1 Click the application button  **> Import > Viewpoints** .
- 2 In the **Import** dialog box, locate the desired viewpoints XML file.



3 Click **Open**.

Clash Test Files

Clash tests can be imported into the **Clash Detective** tool to set up pre-defined, generic clash tests. For more information, see [Import Clash Tests](#) (page 688).

Export Files

3D DWF/DWFX Format

You can export the current 3D model as a DWF or DWFX file.

DWF files are highly compressed and retain detailed design information and scale. DWFX files include additional information to display design data in the Microsoft XPS Viewer. As such, DWFX files are larger than corresponding DWF files.

3D DWF/DWFX file export supports:

- All geometry
- All materials
- Per-vertex colors
- Properties (where available)

NOTE 2D DWF/DWFX file export is not currently supported.

To export a 3D DWF/DWFX file

- 1 Click **Output** tab ► **Export Scene** panel ► **3D DWF/DWFX**  .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

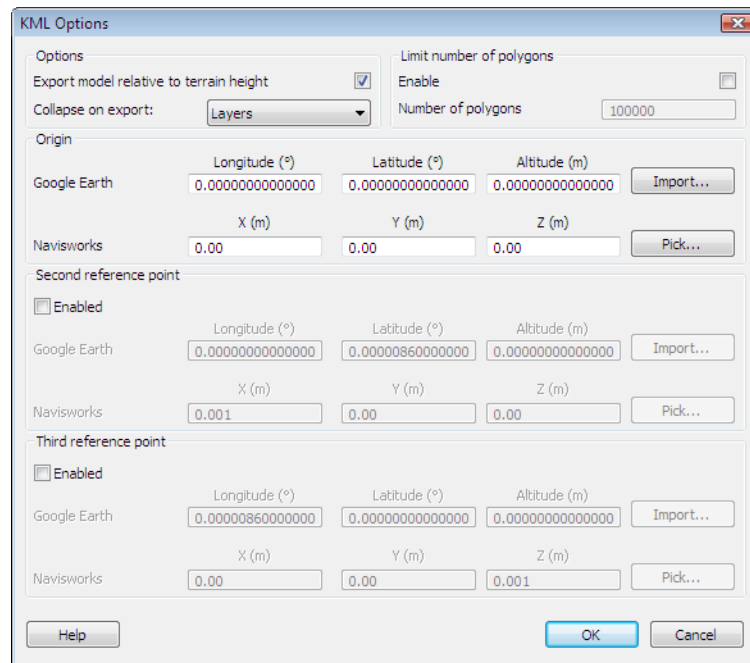
Google Earth KML Format

Google Earth KML files can be exported from Autodesk Navisworks. The exporter creates a compressed KML file with the extension .kmz and supports the export of:

- Triangles
- Lines
- Materials (color and flat transparency only)
- Viewpoints (adjustments may occur due to Google Earth limitations)
- Model Hierarchy
- Hyperlinks (currently only URLs work correctly in Google Earth)

To export to Google Earth KML files

- 1 Click **Output** tab ► **Export Scene** panel ► **Google Earth KML**  .
- 2 In the **KML Options** dialog box, select the **Export Model Relative to Terrain Height** check box to put **Google Earth** in a mode where all heights are measured from the surface of the ground. When this is check box is clear all heights are measured from sea-level.



NOTE When positioning a model relative to sea level, the height of the **Google Earth** reference points must also be measured relative to sea level. When positioning a model relative to the ground, the **Google Earth** reference points must be measured relative to the ground.

Google Earth always places new placemarks at an altitude of zero, irrespective of whether that is relative or absolute.

- 3 The **Collapse on Export** drop-down list allows different levels of collapsing parts of the model hierarchy in the exported file. Choose from:
 - **None** - ensures the whole hierarchy is exported
 - **All Objects** - collapses everything into one node
 - **Files** - collapses each file into one node
 - **Layers** - collapses each layer into one node
- 4 Select the **Enable** check box in the **Limit Number of Polygons** area to restrict the amount of geometry exported into the output file. Geometry is selected on the basis of taking the most obvious objects in preference to the fine detail. If the exported file is too large for **Google Earth** to display, try enabling this and reducing the number of polygons.

Clicking the **Home** tab, sliding out the **Project** tab, and clicking **Scene**

Statistics  in Autodesk Navisworks shows the number of triangles and/or lines in the current project. By setting a polygon limit you are choosing to export only some of these items.

- 5 The **Origin** position values are the first pair of reference points on the **Google Earth** surface, and must always be defined. The Autodesk Navisworks reference point will always be positioned to exactly overlay the **Google Earth** reference point.

Second and **Third** reference points can be used, and if enabled then the position and orientation of the model can be more accurately defined.

- 6 Use the **Import** buttons to read-in saved placemark locations from KML files exported from **Google Earth**.

The **Origin Import** button differs slightly from the other two; if the KML file contains multiple placemarks, this button will offer the user the choice of importing second and third reference points if available. The other two buttons will only import a single reference point.

- 7 The **Pick** buttons allow the reference point locations to be selected in the **Scene View**.

These points must be visible in the **Scene View** prior to exporting, as once the **KML Options** dialog box is open you will not be able to navigate before picking.

TIP You can click **View** tab ➤ **Scene View** panel ➤ **Split View**

drop-down, and select **Split Vertical**  or **Split Horizontal**  to split the **Scene View** into separate views of each reference point.

- 8 Click **OK**.
- 9 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 10 Click **Save**.

Autodesk FBX Format

FBX files can be exported from Autodesk Navisworks. The exporter creates an FBX file with the extension .fbx and supports the export of:

- Triangles
- Lines

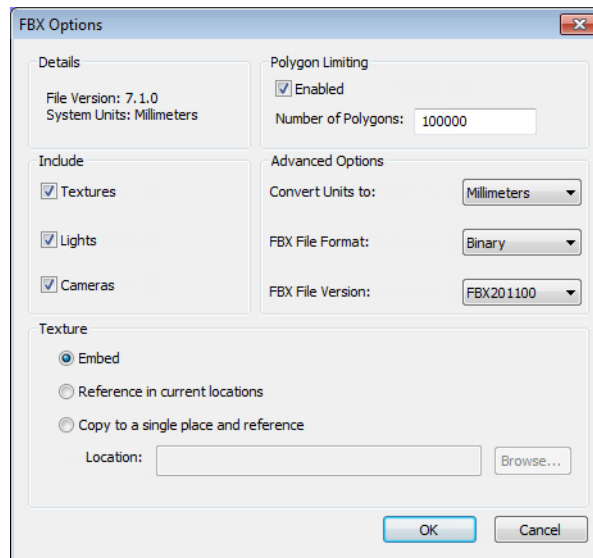
- Materials (color, flat transparency, and wrapped image texture only)
- Viewpoints
- Lights
- Model Hierarchy

NOTE FBX does not support point cloud geometry.

To export an FBX file

- 1 In Autodesk Navisworks application, click **Output** tab ► **Export Scene** panel ► **FBX** .
- 2 In the **FBX Options** dialog box, select the **Enable** check box in the **Polygon Limiting** area to restrict the amount of geometry exported into the output file.

Geometry is selected on the basis of taking the most obvious objects in preference to the fine detail. If the exported file is too large, try enabling this feature and reducing the number of polygons. Clicking the **Home** tab, sliding out the **Project** panel, and clicking **Scene Statistics**  in Autodesk Navisworks shows the number of triangles and lines in the current project. By setting a polygon limit you are choosing to export only some of these items.
- 3 Select the desired options in the **Include** area.
 - **Textures.** Select this check box to include textures in the FBX file.
 - **Lights.** Select this check box to include lights in the FBX file.
 - **Cameras.** Select this check box to include cameras in the FBX file.
- 4 Select the desired **Advanced Options** for the FBX file.
 - **Convert Units To.** Specifies the units that are used in exported FBX file.
 - **FBX File Format** Adjusts the format of exported FBX file (for example, ASCII).
 - **FBX File Version.** Selects the version of exported FBX file (for example, FBX201100).



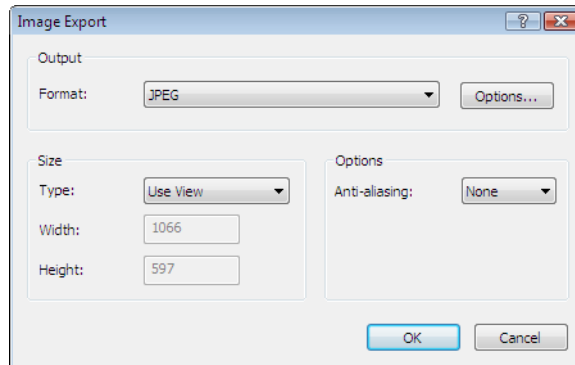
- 5 Click **OK**.
- 6 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 7 Click **Save**.

Export Images and Animations

Export an Image

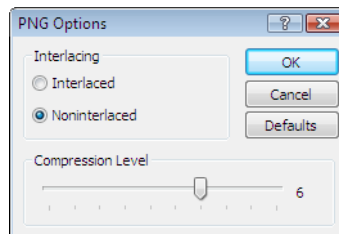
To export an image as a bitmap, PNG, or JPEG file

- 1 Display the view you want to export in the **Scene View**, and click **Output** tab ► **Visuals** panel ► **Image** .
- 2 In the **Image Export** dialog box, select the **Format** of the image you want to export.

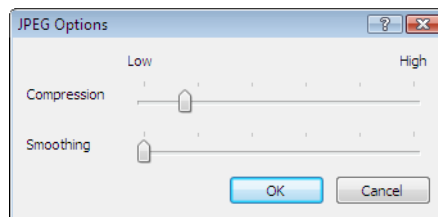


- 3 Use the **Size** and **Options** area to set the image size. See [Image Export Dialog Box](#) (page 760).

For PNG files, click the **Options** button, and use the **PNG Options** dialog box to specify the **Interlacing** and **Compression** settings.



For JPEG files, click the **Options** button, and use the **JPEG Options** dialog box to specify the **Compression** and **Smoothing** settings.



- 4 Click **OK**.
- 5 In the **Save As** dialog box, enter a new filename and location, if you want to change from those suggested.
- 6 Click **Save**.

Export a Rendered Image

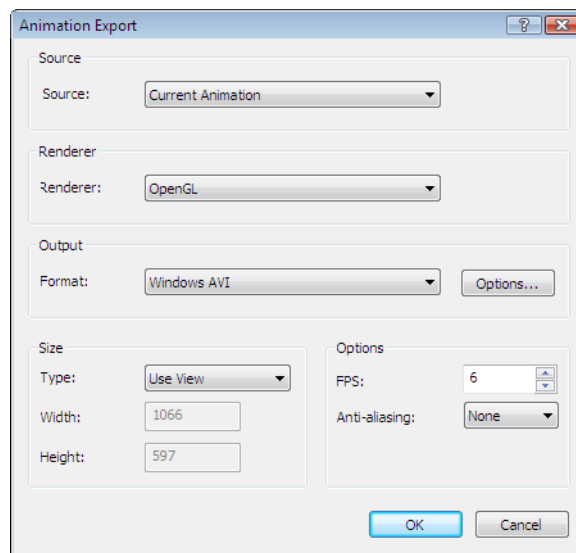
Scenes rendered in **Presenter** can be exported out as images, so they can be used in presentations, on websites, in print, and so on. For more information, see [Photo-Realistic Scene Rendering](#) (page 553).

NOTE This option is only available for 3D models.

Export an Animation

To export an animation to an AVI file, or a sequence of image files

- 1 Click **Output** tab ► **Visuals** panel ► **Animation**  .
The **Animation Export** dialog box opens.



- 2 To export the currently selected viewpoint animation, select **Current Animation** in the **Source** box.
To export the currently selected object animation, select **Current Animator Scene** in the **Source** box.
To export a **TimeLiner** sequence, select **TimeLiner Simulation** in the **Source** box.

- 3 Set up the rest of the boxes in the **Animation Export** dialog box, and click OK. For more information, see [Animation Export Dialog Box](#) (page 737).
- 4 In the **Save As** dialog box, enter a new filename and location, if you want to change from those suggested.
- 5 Click **Save**.

Piranesi EPix Format

This option is only available for 3D models.

To export an EPX file for rendering in Informatix's Piranesi

- 1 **Output** tab ► **Visuals** panel ► **Piranesi EPix** .
- 2 In the **Piranesi EPix** dialog box, click the **Browse** button to locate a destination and enter a new filename to export, if you want to change from the existing filename and location.
- 3 Select the sizing options for the file to be exported:
 - **Type** - sets the size of the exported image as follows:
 - Explicit.** Gives you full control of the width and height (the dimensions are in pixels).
 - Use Aspect Ratio.** Enables you to specify the height. The width is automatically calculated from the aspect ratio of your current view.
 - Use View.** Uses the width and height of your current view.
 - **Width** - enables you to enter the width in pixels, when available.
 - **Height** - enables you to enter the height in pixels, when available.
- 4 Select the desired value in the **Anti-Aliasing** box. This option applies to OpenGL renderer only. Anti-aliasing is used to smooth the edges of the exported images. The higher the number, the smoother the image, but the longer they take to export. 4x is adequate for most situations.
- 5 Click **Save**.

Current Search Criteria

The search criteria specified in the **Find Items** window can be exported to an XML file. This can then be imported into other Autodesk Navisworks sessions. For example, if you have specified a complicated search criteria, containing various logic statements, that relates to all projects you work on, then this feature allows you to specify it once and use it on all projects.

For more information, see [Find Items Window](#) (page 349).

To export current search criteria

- 1 Click **Output** tab ► **Export Data** panel ► **Current Search** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

Search Set Files

Saved search sets can be exported from Autodesk Navisworks as an XML file. These can then be imported into other Autodesk Navisworks sessions and re-used. For example, if you have a number of generic searches that you perform on all of your projects, this feature allows you to specify the searches once and use them on all projects.

For more information see, [Create and Manage Selection and Search Sets](#) (page 362).

To export search sets

- 1 Click **Output** tab ► **Export Data** panel ► **Search Sets** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

Viewpoints Files

Viewpoints can be exported from Autodesk Navisworks to an XML file.

These viewpoints contain all associated data, including camera positions, sections, hidden items and material overrides, redlines, comments, tags and collision detection settings.

Once the viewpoint data is exported to this text-based file format, it can either be imported into other Autodesk Navisworks sessions, or it can be accessed and used in other applications. For example, you may want to set up the same viewpoints in your CAD application.

To export viewpoints

- 1 Click **Output** tab ► **Export Data** panel ► **Viewpoints** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

Viewpoints Report

An HTML file can be exported containing a JPEG of all of the saved viewpoints and associated data, including camera position and comments.

NOTE To customize the appearance or layout of the HTML file, you will need to edit the `viewpoints_report.xml` file. The installed file is located in the stylesheets subdirectory of the Autodesk Navisworks install directory. You can copy the edited file to the stylesheets subdirectory of any of the Autodesk Navisworks search directories. For more information, see [Search Directories](#) (page 159).

To export viewpoints report

- 1 Click **Output** tab ► **Export Data** panel ► **Viewpoints Report** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

TimeLiner CSV

You can export CSV data from **TimeLiner**. Data is exported in default order, without taking account of **TimeLiner** column order or selection.

NOTE When exporting a CSV from **TimeLiner**, the hierarchy of tasks is not represented. All available tasks are exported without any hierarchical structure. This means that collapsing/expanding task nodes in the **TimeLiner** grid does not affect whether or not tasks are output to CSV.

To export CSV data from TimeLiner

- 1 Click **Output** tab ► **Export Data** panel ► **TimeLiner CSV** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

NOTE Exported CSV files always have headers at row 1, which contains the **TimeLiner** column names.

Clash Test Files

Clash tests can be exported from the **Clash Detective** tool for use by other Autodesk Navisworks users. For more information, see [Export Clash Tests](#) (page 689).

PDS Tag Files

To export PDS tags

- 1 Click **Output** tab ► **Export Data** panel ► **PDS Tags** .
- 2 In the **Export** dialog box, enter a new filename and location, if you want to change from those suggested.
- 3 Click **Save**.

Animate Objects

12

In Autodesk Navisworks Manage 2012 you can animate your model and interact with it. For example, you could animate how a crane moves around a site, or how a car is assembled or dismantled, and so on. With a few mouse clicks, you can also create interaction scripts, which link your animations to specific events, such as “On Key Press” or “On Collision”. So, for example, a conveyor belt will move when you press a button on your keyboard, the doors will open as you approach them in your model.

Scope

Autodesk Navisworks Manage 2012 supports both object animation and object interaction as follows:

- Animation specified by keyframes with linear interpolation between keyframes (similar to viewpoints animation).
- Camera, section plane, object animation.
- Multiple independently moving objects independently started.
- Multiple animations of the same object in the same scene (for example, moving a crane, and then lifting its arm).
- Simple scripts (for example, to open and close a specified door).
- Linking to **TimeLiner** to trigger off independent animations as tasks start.
- Linking to **TimeLiner** with different **TimeLiner** tasks play to specific points in an animation (for example, have one animation that moves crane between all points on a site, any task can trigger animation to move from current point to desired point).
- Linking to **Clash Detective** to run clash tests at specific steps through an animation to check for animated versus animated or animated versus static object clashes.

The following is not currently supported in Autodesk Navisworks Manage 2012:

- Animation specified by anything other than objects, for example spline or path.
- Animation of lights.

- Moving a whole object and then creating an independent trigger to move part of object (for example, move whole crane and have button that triggers arm moving at any point).
- Opening/closing of multiple doors without having to create a separate script and animation for each door. Currently there are no cut-and-paste capabilities in scripting.
- Linking to **TimeLiner** to have different **TimeLiner** tasks trigger animations that depend on each other (for example, one task moves crane to a specific point, another task moves arm of crane).
- Real-time collision checking where you turn it on, play an animation and have animation stop when moving object first hits something.

Overview of the Animator Tool

The **Animator** and **Scripter** windows are the two dockable windows that are used to create and edit object animation in Autodesk Navisworks Manage 2012.

Animator Window

Use the **Animator** window to create animated objects in your model.

Ribbon: Animation tab ► Create panel ► **Animator** 

 **Menu:** Classic user interface: **Tools** ► **Animator**

 **Toolbar:** Classic user interface: Workspace ► Animator 

The Animator Toolbar

The Animator toolbar is located at the top of the Animator window.

Use this toolbar to create, edit and play animations.

Control	Purpose
	Puts Animator into translation mode. The Translation gizmo is displayed in the Scene View , and enables you to modify

Control	Purpose
	the position of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar.
	Puts Animator into rotation mode. The Rotation gizmo is displayed in the Scene View , and enables you to modify the rotation of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar.
	Puts Animator into scale mode. The Scaling gizmo is displayed in the Scene View , and enables you to modify the size of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar.
	Puts Animator into color mode. A color palette is shown in the Manual Entry bar, and enables you to modify the color of the geometry objects.
	Puts Animator into transparency mode. A transparency slider is shown in the Manual Entry bar, and enables you to modify the transparency of the geometry objects.
	Takes a snapshot of the current change to the model as a new keyframe in the timeline view.
	Enables/disables snapping. Snapping only comes into effect when moving objects by dragging the gizmos in the Scene View , and has no effect on numerical entry or keyboard control.

Control	Purpose
	Selects the active scene.
	Controls the current position of the time slider in the timeline view.
	Rewinds the animation back to the beginning.
	Rewinds one second.
	Plays the animation backward from the end point to the start, and then stops. This does not alter the direction that the animated elements face.
	Pauses the animation. To continue playing, click Play again.
	Stops and rewinds the animation to the beginning.
	Plays the animation forward from the starting point to the end.
	Plays the animation forward one second.
	Fast forwards the animation to the end.

See also:

[Gizmos](#) (page 160)

The Animator Tree View

The **Animator** tree view lists all scenes and scene components in a hierarchical list view.

Use it to create and manage animation scenes.

Hierarchical List

You can use the **Animator** tree view to create and manage your animation scenes. The scene trees display the scene components such as animation sets, cameras, and section planes in a hierarchical structure.

To work with an item in the tree view, you must first select it.

Selecting a scene component in the tree view selects all the elements contained within that component in the **Scene View**. For example, selecting an animation set in the tree view automatically selects all geometry objects contained within that animation set.

You can quickly copy and move items in the tree view by dragging them. To do this, click the item you want to copy or move, hold down the right mouse button, and drag the item to the desired location. When the mouse pointer changes to an arrow, release the right mouse button to display a shortcut menu. Click **Copy Here** or **Move Here**, as appropriate.

Shortcut Menu

You can display a shortcut menu for any item in the tree by right-clicking an item. The following commands are available on the shortcut menu whenever they are applicable:

Command	Purpose
Add Scene	Adds a new scene to the tree view.
Add Camera	Adds a new camera to the tree view.
Add Animation Set	Adds an animation set to the tree view.
Update Animation Set	Updates the selected animation set.
Add Section Plane	Adds a new section plane to the tree view.

Command	Purpose
Add Folder	Adds a folder to the tree view. Folders can hold scene components and other folders.
Add Scene Folder	Adds a scene folder to the tree view. Scene folders can hold scenes and other scene folders. When you add a scene folder, if you do this while having an empty scene folder selected, Autodesk Navisworks creates the new scene folder at the very top of the tree, otherwise it creates it beneath your current selection.
Active	Enables or disables a scene component.
Loop	Selects loop mode for scenes and scene animations. The animation plays forward to the end, then restarts again from the beginning, looping indefinitely.
Ping-Pong	Selects ping-pong mode for scenes and scene animations. The animation plays forward to the end, and then plays backward to the beginning. Unless loop mode is also selected, this will only happen once.
Infinite	Selects infinite mode; it only applies to scenes, and will make the scene play indefinitely (that is until Stop  is clicked).
Cut	Cuts the selected item in the tree to clipboard.
Copy	Copies the selected item in the tree to clipboard.
Paste	Pastes the item from clipboard into the new location.

Command	Purpose
Delete	Deletes the selected item from the tree.

Icons

Icon	Purpose
	Opens a shortcut menu that enables you to add new items to the tree view, such as Add Scene , Add Camera and so on.
	Deletes the currently selected item in the tree view. NOTE If you accidentally delete an item, click Undo  on the Quick Access toolbar to restore it.
	Moves the currently selected scene up in the tree view.
	Moves the currently selected scene down in the tree view.
	Zoom in on the timescale bar. The actual value is displayed in the Zoom box to the right.
	Zoom out on the timescale bar. The actual value is displayed in the Zoom box to the right.

Check Boxes

Use the check boxes in the scene view to control whether the corresponding item is active, whether it loops, ping-pongs, and if it should run infinitely.

Active This check box is only available for scene animations. Check this check box to make the animation active in the scene. Only active animations will play.

NOTE To make a scene active, you need to select it in the **Scene Picker** on the **Animator** toolbar.

Loop This check box is available for scenes and scene animations. It enables you to control the playback mode. Check this check box to use loop mode. When the animation reaches the end, it will reset back to the start and run again.

P.P. This check box is available for scenes and scene animations. It enables you to control the playback mode. Check this check box to use ping-pong mode. When the animation reaches the end, it will run backward until it reaches the start. Unless loop mode is also selected, this will only happen once.

Infinite This check box is only available for scenes. Check this check box to make the scene play indefinitely (that is, until **Stop**  is clicked). If this check box is unchecked, the scene will play until its end point is reached.

NOTE When a scene is set to **Infinite**, it can not also loop or ping-pong; so if you check this check box, the **Loop** and **P.P.** check boxes are not available for your scene.

The Animator Timeline View

The timeline view shows the timelines with keyframes for animation sets, cameras, and section planes in your scenes.

Use it to visualize and edit the animations.

Timescale Bar

At the top of the timeline view is the timescale bar in seconds. All timelines start at 0. Right-clicking the timescale bar opens a shortcut menu.

Use the  and  icons at the bottom of the **Animator** tree view to zoom in and out on the timescale bar.

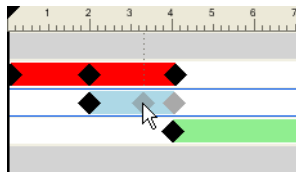
The default timescale shows around 10 seconds of animation on a standard screen resolution, zooming in and out has an effect of doubling or halving the visible area. So, for example, zooming in shows around 5 seconds of animation, and zooming out shows around 20 seconds.

TIP You can also zoom in and out with mouse wheel while hovering over the timeline.

Another way of changing the timescale is to use the **Zoom** box. For example, type in “1/4”, and press Enter to quarter the visible area. The value you enter is halved when you zoom in, and doubled when you zoom out. You can return to the default timescale by deleting the **Zoom** box value, and pressing Enter.

Keyframes

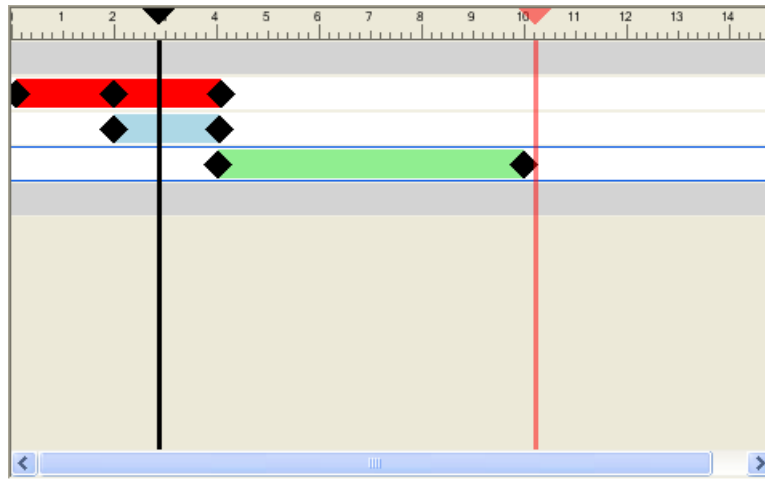
The keyframes are shown as black diamonds in the timelines. You can change the time when a keyframe occurs by dragging the black diamond left or right in the timeline view. As you drag a keyframe, it changes the color from black to a lighter grey.



Left-clicking a keyframe moves the time slider to that position. Right-clicking a keyframe opens a shortcut menu.

Animation Bars

The colored animation bars are used to visualize the keyframes in a timeline, and cannot be edited. Each animation type is shown in a different color, and the animation bars for the scenes are grey. Typically, the animation bar ends with the last keyframe. If an animation bar carries on in a faded color after the last keyframe, this indicates that the animation will play indefinitely (or, loop animation).



Sliders

You can use two sliders in the timeline view:

Time Slider The black vertical line is the time slider, representing the current position in the playback. This can be adjusted by either using the VCR controls on the **Animator** toolbar, by dragging the time slider left or right in the timeline view, or by selecting the time slider in the timeline view and using the left and right arrow keys. As the time slider is moved, the model in the **Scene View** updates to reflect movements caused by keyframes in the current scene as if the animation were playing back.

End Slider The red vertical line is the end slider, representing the end point of the current active scene. It is hidden if you selected the **Infinite** check box for the current scene in the tree view. By default, the end slider is set to the last keyframe in the scene and can't be moved. You can switch on manual control of the end slider in a scene by right-clicking the end slider, and choosing **Manually Position Endbar** from the shortcut menu. Then, manually adjust the end slider, by dragging it left or right. This end marker will be used as the animation end point when scenes are set to loop or ping-pong.

Shortcut Menu

Right-clicking the timescale bar opens a shortcut menu that allows you to adjust the sliders.

- **Move Time Here** - moves the time slider into the current mouse pointer position.
- **Move Scene End Here** - moves the end slider into the current mouse pointer position.

NOTE This option is not available until you select the **Manually Position Endbar** option.

- **Reset Endbar to the Scene End** - moves the end slider to its original position.
- **Manually Position Endbar** - switches on manual control of the end slider in a scene.

Right-clicking a keyframe opens the shortcut menu with the following options:

- **Edit** - enables you to edit the selected keyframe.
- **Go to Keyframe** - moves the black time slider to this keyframe.
- **Cut, Copy, Paste** - standard cut, copy, and paste commands. They only work inside the selected timeline.
- **Delete** - deletes the selected keyframe.
- **Interpolate** - determines whether Autodesk Navisworks automatically interpolates between the current and the last keyframes. This is the default option. When disabled, there will be no gradual transition between the two keyframes; instead the animation will instantly jump to the position/view of the second keyframe when it's reached. Also, there will be no colored animation bar between the keyframes.

NOTE If you right-click an area on an animation bar without a keyframe in it, the shortcut menu will contain only the **Paste** option; to use it, you must have the keyframe data on the clipboard.

See also:

[Work with Keyframes](#) (page 538)

The Manual Entry Bar

The optional **Manual Entry** bar is located at the bottom of the **Animator** window, and enables you to manipulate geometry objects by typing in numerical values, instead of using gizmos in the **Scene View**.

The contents of the **Manual Entry** bar change depending on the button last selected from the **Animator** toolbar.

You can toggle the **Manual Entry** bar on and off by using the [Options Editor](#) (page 827).

Translate

X, Y, Z Enter the X, Y, and Z coordinate values to position the selected objects.

Rotate

X, Y, Z Enter the degrees of rotation around X, Y, and Z axes to move the selected objects into this position.

cX, cY, cZ Enter the X, Y, and Z coordinate values to move the origin (or center point) of the rotation into this position.

oX, oY, oZ Enter the degrees of rotation around X, Y, and Z axes to modify the orientation of the rotation.

NOTE 0,0,0 represents world up.

Scale

X, Y, Z Enter the scaling factor around X, Y, and Z axis; 1 being the current size, 0.5 half, 2 being double and so on.

cX, cY, cZ Enter the X, Y, and Z coordinate values to move the origin (or center point) of the scaling into this position.

Color

Color This check box indicates whether or not clicking **Capture Keyframe**  will record color change in the keyframe.

Unchecking this check box resets the color back to its original state.

By default, Autodesk Navisworks unchecks this check box every time you create an animation set. Within one and the same animation set, once this check box is selected, it stays selected, until you manually uncheck it.

R, G, B Enter the Red, Green, and Blue values for the new color.

 If you don't want to enter the Red, Green, and Blue values manually, click this button, and select the desired color.

Transparency

Transparency This check box indicates whether or not clicking **Capture Keyframe**  will record transparency change in the keyframe.

Unchecking this check box resets the transparency back to its original state.

By default, Autodesk Navisworks unchecks this check box every time you create an animation set. Within one and the same animation set, once this check box is selected, it stays selected, until you manually uncheck it.

% Enter the value to adjust the transparency level (0 to 100%). A higher value makes the element more transparent; a lower value makes the element more opaque.

 If you don't want to enter the transparency value manually, use this slider to adjust the transparency level.

See also:

[Gizmos](#) (page 160)

Scripter Window

Use the **Scripter** window to add interactivity to animated objects in your model.

Ribbon: **Animation** tab ► **Script** panel ► **Scripter** 

 **Menu:** Classic user interface: Tools ► Scripter

 **Toolbar:** Classic user interface: Workspace ► Scripter 

The Scripter Tree View

The **Scripter** tree view contains all scripts available in your Autodesk Navisworks file in a hierarchical list view.

Use it to create and manage animation scripts.

NOTE Although you can organize the scripts into folders, this has no effect on the way scripts are executed in Autodesk Navisworks.

Hierarchical List

You can use the **Scripter** tree view to create and manage your scripts.

To work with an item in the tree view, you must first select it.

Selecting a script in the tree view displays the associated events, actions and properties.

You can quickly copy and move items in the tree view by dragging them. To do this, click the item you want to copy or move, hold down the right mouse button, and drag the item to the desired location. When the mouse pointer changes to an arrow, release the mouse button to display a shortcut menu. Click **Copy Here** or **Move Here**, as appropriate.

Shortcut Menu

You can display a shortcut menu for any item in the tree by right-clicking an item. The following commands are available on the shortcut menu whenever they are applicable:

Command	Purpose
Add New Script	Adds a new script to the tree view.
Add New Folder	Adds a folder to the tree view. Folders can hold scripts and other folders.
Rename Item	Enables you to rename the currently selected item in the tree view.
Delete Item	Deletes the currently selected item in the tree view.
Activate	Selects the Active check box for the currently selected item in the tree view. Only active scripts will be executed.

Command	Purpose
Deactivate	Unchecks the Active check box for the currently selected item in the tree view. Only active scripts will be executed.

Icons

Icon	Purpose
	Adds new scripts to the tree view.
	Adds new folders to the tree view.
	Deletes the currently selected item in the tree view. NOTE If you accidentally delete an item, click Undo  on the Quick Access toolbar to restore it.

Check Boxes

Active Use this check box to specify which scripts you want to use. Only active scripts are executed.

If you organized your scripts into folders, you can quickly turn the scripts on and off by using the **Active** check box next to the top-level folder.

The Events View

The **Events** view shows all events associated with the currently selected script.

Use the **Events** view to define, manage, and test events.

Icons

Icon	Purpose
	Adds start events.
	Adds timer events.
	Adds key press events.
	Add collision events.
	Adds hotspot events.
	Adds variable events.
	Adds animation events.
	Moves the currently selected event up in the Events view.
	Moves the currently selected event down in the Events view.
	Deletes the currently selected event in the Events view.

Shortcut Menu

Right-click in the **Events** view to display a shortcut menu. The following commands are available on the shortcut menu whenever they are applicable:

Command	Purpose
Add Event	Enables you to select an event to add.

Command	Purpose
Delete Event	Deletes the currently selected event.
Move Up	Moves the currently selected event up.
Move Down	Moves the currently selected event down.
Brackets	Enables you to select brackets. Choose from (,) , and None .
Logic	Enables you to select logic operators. Choose from AND , and OR .
Test Logic	Tests the validity of your event condition.

The Actions View

The **Actions** view shows the actions associated with the currently selected script.

Use the **Actions** view to define, manage, and test actions.

Icons

Icon	Purpose
	Adds play animation actions.
	Adds stop animation actions.
	Adds show viewpoint actions.
	Adds pause actions.

Icon	Purpose
	Adds send message actions.
	Adds set variable actions.
	Adds store property actions.
	Adds load model actions.
	Moves the currently selected action up in the Actions view.
	Moves the currently selected action down in the Actions view.
	Deletes the currently selected action.

Shortcut Menu

Right-click in the **Actions** view to display a shortcut menu. The following commands are available on the shortcut menu whenever they are applicable:

Command	Purpose
Add Action	Enables you to select an action to add.
Delete Action	Deletes the currently selected action.
Test Action	Executes the currently selected action.
Stop Action	Stops execution of the currently selected action (when you Test Action).
Move Up	Moves the currently selected action up in the Actions view.

Command	Purpose
Move Down	Moves the currently selected action down in the Actions view.

The Properties View

The **Properties** view shows the properties for the currently selected event or action.

Use the **Properties** view to configure the behavior of events and actions in your scripts.

Events Properties

Currently, there are seven event types in Autodesk Navisworks. When an event is added, the **Properties** view displays the properties for that event type. The event properties can be configured immediately, or at a later time.

On Start

You don't need to configure any properties for this event type.

On Timer

Interval (Seconds) Defines the length of time in seconds between timer triggering.

Regularity Specifies the event frequency. Choose from:

- **Once After** - an event happens once only. Use this option to create an event that starts after a certain length of time.
- **Continuous** - an event is continuously repeated at specified time intervals. You can use this, for example, to simulate cyclic work of a factory machine.

On Key Press

Key Click in this box and press the key to link it to your event.

Trigger On Defines how the event is triggered. Choose from:

- **Key Up** - an event is triggered after you press and release the key.

- **Key Down** - an event is triggered as soon as you press the key down.
- **Key Pressed** - an event is triggered while the key is pressed. This option allows you to use a key press event together with Boolean operators. For example, you can AND this event to a timer event.

On Collision

Selection to Collide With Click the **Set** button, and use the shortcut menu to define the collision objects:

- **Clear** - clears your currently selected collision objects.
- **Set From Current Selection** - sets the collision objects to your current object selection in the **Scene View**.

NOTE This option is not available until you make a selection in the **Scene View**.

- **Set From Current Selection Set** - sets the collision objects to your current search set or selection set.

Show This is a read-only box, showing the number of geometry objects selected as collision objects.

Include the Effects of Gravity Check this check box if you want to include gravity in collision. If this option is used, hitting floor when walking across it, for example, will trigger your event.

On Hotspot

Hotspot Defines the hotspot type. Choose from:

- **Sphere** - a simple sphere from a given point in space.
- **Sphere on Selection** - a sphere around a selection. This option doesn't require you to define the given point in space. This hotspot will move as the selected objects move in the model.

Trigger When Defines how the event is triggered. Choose from:

- **Entering** - an event is triggered when you enter the hotspot. This is useful for opening doors, for example.
- **Leaving** - an event is triggered when you leave the hotspot. This is useful for closing doors, for example.
- **In Range** - an event is triggered when you are inside the hotspot. This option allows you to use a hotspot event together with Boolean operators. For example, you can AND this event to a timer event.

Hotspot Type

Position The position of the hotspot point. If the chosen hotspot is **Sphere on Selection**, this property is not available.

Pick Enables you to pick the position of the hotspot point. If the chosen hotspot is **Sphere on Selection**, this button is not available. Click the **Pick** button, and then click a point for the hotspot in the **Scene View**.

Selection Click the **Set** button, and use the shortcut menu to define the hotspot objects:

NOTE If the chosen hotspot is **Sphere**, this button is not available.

- **Clear** - clears the current selection.
- **Set From Current Selection** - the hotspot is set to your current object selection in the **Scene View**.

NOTE This option is not available until you make a selection in the **Scene View**.

- **Set From Current Selection Set** - the hotspot is set to your current search set or selection set.

Show This is a read-only box, which displays the number of geometry objects linked to the hotspot. If the chosen hotspot is **Sphere**, this property is not available.

Radius (m) Radius of hotspot.

On Variable

Variable The alphanumeric name of the variable to be evaluated.

Value An operand to use. Enter a value to be tested against your variable. Alternatively, enter a name of another variable. Its value will be tested against the value in your variable.

The following rules apply:

- If you enter a number (for example 0, 400, 5.3), the value is treated as a numeric value. If it has a decimal place, the floating-point formatting is preserved up to the user-defined decimal places.
- If you enter an alphanumeric string between single or double quotation marks, such as 'testing' or "hello", the value is treated as a string.
- If you enter an alphanumeric string without single or double quotation marks, such as counter1 or testing, the value is treated as another variable.

If this variable has never been used before, it's assigned a numerical value of 0.

- If you enter the word true or false without any quotes, the value is treated as a Boolean (true = 1, false = 0).

Evaluation Operators used for variable comparison. You can use any of the following operators with numbers and Boolean values. However, comparing strings is limited to the “Equal To” and “Not Equal To” operators only.

- **Equal To**
- **Not Equal To**
- **Greater Than**
- **Less Than**
- **Greater Than or Equal To**
- **Less Than or Equal To**

On Animation

Animation Selects the animation that triggers the event. If you don't have any object animation in your Autodesk Navisworks file, this property is not available.

Trigger On Defines how the event is triggered. Choose from:

- **Starting** - an event is triggered when the animation starts.
- **Ending** - an event is triggered when the animation ends. This is useful for chaining animations together.

See also:

[Event Types](#) (page 543)

Actions Properties

Currently, there are eight action types in Autodesk Navisworks. When an action is added, the **Properties** view displays the properties for that action type. The action properties can be configured immediately, or at a later time.

Play Animation

Animation Selects the animation to play. If you don't have any object animation in your Autodesk Navisworks file, this property is not available.

Pause at End Check this check box if you want the animation to stop at the end. When this check box is unchecked, the animation snaps back to the starting point when it ends.

Starting At Defines the starting position of the animation playback. Choose from:

- **Start** - the animation plays forward from the beginning.
- **End** - the animation plays backward from the end.
- **Current Position** - the animation plays from its current position, if the playback has already started. Otherwise, the animation plays forward from the beginning.
- **Specified Time** - the animation plays from the segment defined in the **Specific Start Time (Seconds)** property.

Ending At Defines the finishing position of the animation playback. Choose from:

- **Start** - the playback ends at the beginning of the animation.
- **End** - the playback ends at the end of the animation.
- **Specified Time** - the playback ends at the segment defined in the **Specific End Time (Seconds)** property.

Specific Start Time (Seconds) The starting position of a playback segment.

Specific End Time (Seconds) The finishing position of a playback segment.

Stop Animation

Animation Selects the animation to stop. If you don't have any object animation in your Autodesk Navisworks file, this property is not available.

Reset To Defines the playback position of the stopped animation. Choose from:

- **Default Position** - the animation is reset to its starting point.
- **Current Position** - the animation remains at the position it was stopped at.

Show Viewpoint

Viewpoint Selects the viewpoint or a viewpoint animation to show. If you don't have any viewpoints in your Autodesk Navisworks file, this property is not available.

Pause

Delay (Seconds) Defines the amount of time delay before the next action in the script is run.

Send Message

Message Defines the message to send to a text file defined in the **Options Editor**.

You can output the **Scripter** variables in your message. To do this, use `%variable_name%` style.

Set Variable

Variable Name The alphanumeric name for the variable.

Value An operand to assign. The following rules apply:

- If you enter a number (for example 0, 400, 5.3), the value is treated as a numeric value. If it has a decimal place, the floating-point formatting is preserved up to the user-defined decimal places.
- If you enter an alphanumeric string between single or double quotation marks, such as 'testing' or "hello", the value is treated as a string.
- If you enter the word true or false without any quotes, the value is treated as a Boolean (true = 1, false = 0).

Modifier Assignment operators for your variable. You can use any of the following operators with numbers and Boolean values. However, using strings is limited to the **Set Equal To** operator only.

- **Set Equal To**
- **Increment By**
- **Decrement By**

Store Property

Selection to Get Property From Click the **Set** button, and use the shortcut menu to define the objects, which are used to get the property from:

- **Clear** - clears the current selection.
- **Set From Current Selection** - the objects are set to your current object selection in the **Scene View**.

NOTE This option is not available until you make a selection in the **Scene View**.

- **Set From Current Selection Set** - the objects are set to your current search set or selection set.

IMPORTANT If your selection contains a hierarchy of objects, the top-level object's property is automatically used. So, for example, if you selected a group called "Wheel", which includes two subgroups called "Rim", and "Tire", only the properties that relate to "Wheel" can be stored.

Variable to Set The name of the variable to receive the property.

Property to Store

Category The property category. The values in this drop-down list depend on the selected objects.

Property The property type. The values in this drop-down list depend on the chosen property **Category**.

Load Model

File to Load The path to the Autodesk Navisworks file that will be loaded to replace the current one.

You may find this useful when you present a selection of animated scenes contained in a range of different model files.

See also:

[Action Types](#) (page 545)

Create Object Animations

An animation is a prepared sequence of changes to the model. The changes you can make in Autodesk Navisworks Manage 2012 are:

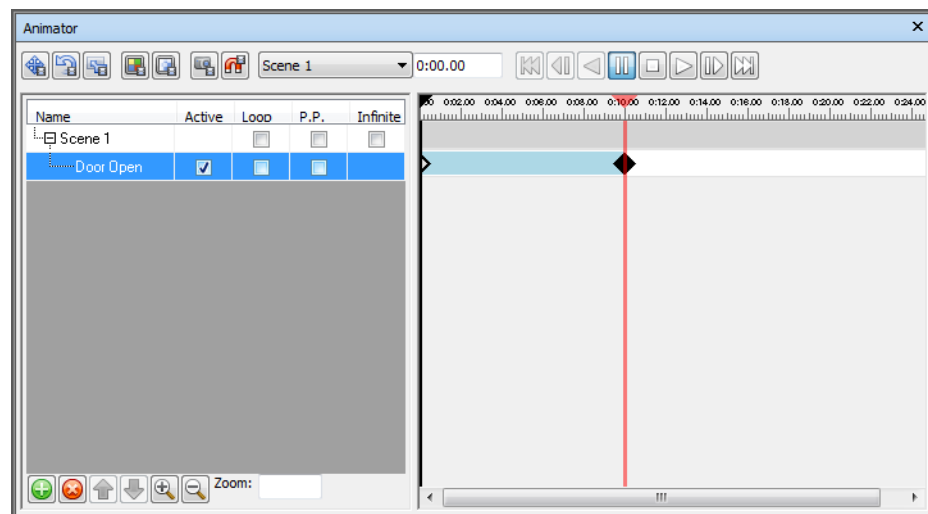
- Manipulating geometry objects by modifying their position, rotation, size, and appearance (color, and transparency). This type of change is referred to as an animation set.
- Manipulating viewpoints by using different navigation tools (such as orbiting or flying), or by using existing viewpoint animations. This type of change is referred to as a camera.
- Manipulating cross-sectional cuts of your model either by moving the section planes or a section box. This type of change is referred to as a section plane set.

The **Animator** window is a floating window that enables you to add animations to your model.

Ribbon: Animation tab ► Create panel ► Animator 

Menu: Classic user interface: Tools ► Animator

Toolbar: Classic user interface: Workspace ► Animator 



The **Animator** window contains the following components: [the toolbar](#) (page 502), [the tree view](#) (page 505), [the timeline view](#) (page 508), and [the manual entry bar](#) (page 512).

Work with Animation Scenes

Scenes act as containers for your object animations.

Each scene can contain the following components:

- One or more animation sets
- A single camera animation
- A single section plane set animation

The scenes and the scene components can be grouped into folders. This has no effect on playback, except that the contents of a folder can be easily switched on or off to save time.

There are two types of folders:

- **Scene folders** - to hold scenes and other scene folders.
- **Folders** - to hold scene components and other folders.

To add an animation scene

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator** .
- 2 Right-click in the **Animator** tree view, and click **Add Scene** on the shortcut menu.
- 3 Click the default scene name, and type in a new name.

TIP Use names that can help you to identify your scenes in future.

Pointing device: Click , and click **Add Scene** on the shortcut menu.

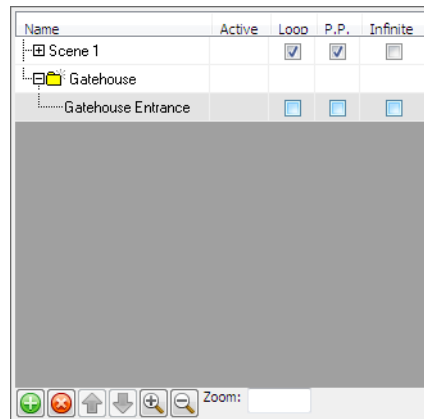
To delete an animation scene

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator** .
- 2 Select the scene you want to delete, and click .

NOTE Deleting a scene also deletes all of its components. If you accidentally delete an item, click **Undo**  on the **Quick Access** toolbar to restore it.

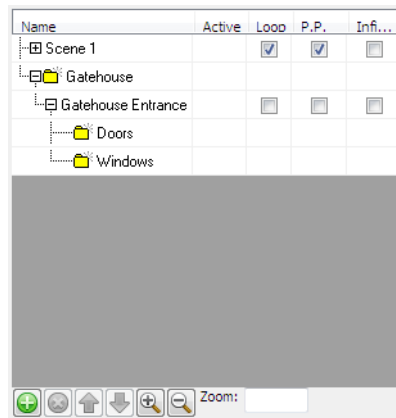
To organize scenes into scene folders

- 1 If the **Animator** window is not already open, click **Animation** tab
➤ **Create** panel ➤ **Animator** .
- 2 Right-click in the **Animator** tree view, and click **Add Scene Folder** on the shortcut menu.
- 3 Click the default folder name, and type in a new name.
TIP Use names that can help you to identify your scenes in future.
- 4 Select the scene you want to add to your new folder. Hold down the left mouse button, and drag the mouse to the folder name. When the mouse pointer changes to an arrow, release the mouse button to drop the scene into the folder.



To organize scene components into folders

- 1 If the **Animator** window is not already open, click **Animation** tab
➤ **Create** panel ➤ **Animator** .
- 2 To add a subfolder to a scene, right-click it, and click **Add Folder** on the shortcut menu.
To add a subfolder to a scene component, right-click it, and click **Add Folder** on the shortcut menu.



- 3 To move a subfolder, right-click it, and click **Cut** on the shortcut menu. Right-click the new location, and click **Paste** on the shortcut menu.
- 4 To rename a folder, click it, and type a new name.

TIP Use names that can help you to identify your scene components in future.

Work with Animation Sets

An animation set contains a list of geometry objects to be animated, and a list of keyframes that describe how it is to be animated.

Your scene can have as many animation sets as you want, and you can also have the same geometry objects in different animation sets within the same scene. The order of animation sets within a scene is important, and can be used to control the final object position when the same object is used in multiple animation sets.

Add Animation Sets

An animation set can be based on the current selection in the **Scene View**, or on the current selection set or current search set.

When you add an animation set based on a selection set, the contents of the animation set is automatically updated if the contents of the source selection set change.

When you add an animation set based on a search set, the contents of the animation set is updated each time the model changes to include everything in the search set.

NOTE Any changes to search/selection sets during the animation playback are ignored.

If the model changes so that objects in a particular animation are missing, they are automatically removed from the animation set when the corresponding NWD or NWF file is resaved.

Finally, if the selection or search sets are deleted rather than being lost, the corresponding animation set becomes a static selection of objects based on what it last contained.

To add an animation set based on current selection

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator** .
- 2 Select the desired geometry objects in the **Scene View**, or from the **Selection Tree**.
- 3 Right-click the scene name, and click **Add Animation Set ➤ From Current Selection** on the shortcut menu.
- 4 If desired, type a name for the new animation set and press Enter.

To add an animation set based on current search set or selection set

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator** .
- 2 Select the desired search set or selection set from the **Sets** window.
- 3 Right-click the scene name, and click **Add Animation Set ➤ From Current Search/Selection Set** on the shortcut menu.
- 4 If desired, type a name for the new animation set and press Enter.

Update Animation Sets

An animation set can be manually updated.

You can modify your current selection in the **Scene View**, or the current selection set or current search set, and change the contents of your animation set to reflect this.

NOTE The keyframes are not affected by this operation.

To update an animation set based on current selection

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator** .
- 2 Select the desired geometry objects in the **Scene View**, or from the **Selection Tree**.
- 3 Right-click the scene name, and click **Update Animation Set > From Current Selection** on the shortcut menu.

To update an animation set based on current search set or selection set

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator** .
- 2 Select the desired search set or selection set from the **Sets** window.
- 3 Right-click the scene name, and click **Update Animation Set > From Current Search/Selection Set** on the shortcut menu.

Manipulate Geometry Objects

You can modify position, rotation, size, color and transparency of geometry objects in animation sets, and capture these changes in keyframes.

All object manipulation is carried out the in the **Scene View**.

When you manipulate geometry objects by changing their position, rotation or size you can use snapping to control the precision of your operations in the **Scene View**.

When you select animation sets in the **Animator** tree view, they are also highlighted in the **Scene View**. To get a clearer view of an object as you animating it, you can use the **Options Editor** to adjust the way in which the current selection is highlighted.

See also:

[The Manual Entry Bar](#) (page 512)

[Gizmos](#) (page 160)

To capture object movement

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator** .
- 2 Select the desired animation set in the **Animator** tree view.
The corresponding geometry objects are highlighted in the **Scene View**.
TIP To get a better view, change the way the objects are highlighted.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial object state.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Click **Translate Animation Set**  on the **Animator** toolbar.
- 6 Use the **Translation** gizmo to change the position of the selected objects.
- 7 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

To capture object rotation

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator** .
- 2 Select the desired animation set in the **Animator** tree view.
The corresponding geometry objects are highlighted in the **Scene View**.
TIP To get a better view, change the way the objects are highlighted.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial object state.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Click **Rotate Animation Set**  on the **Animator** toolbar.

- 6 Use the **Rotation** gizmo to rotate the selected objects.
- 7 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

To capture scaling changes

- 1 If the **Animator** window is not already open, click **Animation** tab  **> Create panel > Animator** .
- 2 Select the desired animation set in the **Animator** tree view.
The corresponding geometry objects are highlighted in the **Scene View**.
TIP To get a better view, change the way the objects are highlighted.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial object state.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Click **Scale Animation Set**  on the **Animator** toolbar.
- 6 Use the **Scale** gizmo to resize the selected objects.
- 7 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

To capture color changes

- 1 If the **Animator** window is not already open, click **Animation** tab  **> Create panel > Animator** .
- 2 Select the desired animation set in the **Animator** tree view.
The corresponding geometry objects are highlighted in the **Scene View**.
TIP To get a better view, change the way the objects are highlighted.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial object state.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Click **Change Color of Animation Set**  on the **Animator** toolbar.
- 6 Click the **Color** button on the **Manual Entry** bar, and choose the desired color.

- 7 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

To capture transparency changes

- 1 If the **Animator** window is not already open, click **Animation** tab .
 > **Create** panel > **Animator** .
- 2 Select the desired animation set in the **Animator** tree view.
The corresponding geometry objects are highlighted in the **Scene View**.
TIP To get a better view, change the way the objects are highlighted.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial object state.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Click **Change Transparency of Animation Set**  on the **Animator** toolbar.
- 6 Use the **Transparency** slider on the **Manual Entry** bar to adjust how transparent or opaque the selected objects are.
- 7 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

Work with Cameras

A camera contains a list of viewpoints, and an optional list of keyframes to describe how the viewpoints move.

If no camera keyframes are defined, then the scene uses the current views in the **Scene View**. If a single keyframe is defined, the camera moves to that viewpoint, and then remain static throughout the scene. Finally, if multiple keyframes are defined, the camera is animated accordingly.

You can add a blank camera, and then manipulate the viewpoints, or you can copy an existing viewpoint animation straight into your camera.

NOTE Each scene can only have one camera in it.

To add a blank camera

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 Right-click the desired scene name, and click **Add Camera ➤ Blank Camera** on the shortcut menu.
You are now ready to capture camera viewpoints.

To add a camera with existing viewpoint animation

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 Select the desired viewpoint animation from the **Viewpoints** control bar.
- 3 Right-click the desired scene name, and click **Add Camera ➤ From Current Viewpoint Animation** on the shortcut menu.
Autodesk Navisworks automatically adds all necessary keyframes to the timeline view.

To capture camera viewpoints

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 Select the desired camera in the **Animator** tree view.
- 3 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the current viewpoint.
- 4 In the timeline view, move the black time slider to the right to set the desired time.
- 5 Use the buttons on the navigation bar to change your current viewpoint.
Alternatively, select one of the saved viewpoints from the Viewpoints control bar.
- 6 To capture the current object changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

Work with Section Plane Sets

A section plane set contains a list of cross-sectional cuts of your model, and a list of keyframes to describe how they move.

NOTE Each scene can only have one section plane set in it.

See also:

[Gizmos](#) (page 160)

[Sectioning](#) (page 455)

To add a section plane set

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator**  .
- 2 Right-click the desired scene name, and click **Add Section Plane** on the shortcut menu.
 You are now ready to capture cross-sectional cuts.

To capture cross-sectional cuts by moving a section plane

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator**  .
- 2 Select the desired section plane set in the **Animator** tree view.
- 3 Click **Viewpoint** tab > **Sectioning** panel > **Enable Sectioning**  .
 Autodesk Navisworks opens the **Sectioning Tools** tab on the ribbon, and draws a section plane through the model in the **Scene View**.
- 4 Click **Sectioning Tools** tab > **Planes Settings** panel > **Current Plane** drop-down, and select the plane that you want to manipulate.
- 5 Click **Sectioning Tools** tab > **Transform** panel and click the sectioning gizmo you want to use (move  or rotate ). By default, the move gizmo is used.
 Drag the gizmo to adjust the initial position of the plane in the **Scene View**.

- 6 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial position of the section plane.
- 7 In the timeline view, move the black time slider to the right to set the desired time.
- 8 Use the gizmo again to adjust the depth of your cross-section cut.
- 9 To capture the current plane changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

To capture cross-sectional cuts by moving a section box

- 1 If the **Animator** window is not already open, click **Animation** tab
 > **Create** panel > **Animator** .
- 2 Select the desired section plane set in the **Animator** tree view.
- 3 Click **Viewpoint** tab > **Sectioning** panel > **Enable Sectioning** .

Autodesk Navisworks opens the **Sectioning Tools** tab on the ribbon, and draws a section plane through the model in the **Scene View**.

- 4 Click **Sectioning Tools** tab > **Mode** panel > **Box** .
- 5 Click **Sectioning Tools** tab > **Transform** panel and click the sectioning gizmo you want to use (move  , rotate  , or scale ). By default, the move gizmo is used.
 Drag the gizmo to adjust the initial position of the section box in the **Scene View**.
- 6 Click **Capture Keyframe**  on the **Animator** toolbar to create a keyframe with the initial position of the section box.
- 7 In the timeline view, move the black time slider to the right to set the desired time.
- 8 Use the gizmo again to adjust the depth of your cross-section cut.
- 9 To capture the current section box changes in a keyframe, click **Capture Keyframe**  on the **Animator** toolbar.

Work with Keyframes

Keyframes are used to define position and properties of the changes made to the model.

See also:

[Edit Key Frame Dialog Box](#) (page 748)

Capture Keyframes

New keyframes are created by clicking **Capture Keyframe**  on the **Animator** toolbar. Every time you click this button, Autodesk Navisworks adds a keyframe of the currently selected animation set, camera, or section plane set at the current position of the black time slider.

Conceptually, keyframes represent relative translations, rotations and scaling operations from the previous keyframe or, in the case of the first keyframe, the model's starting position.

Keyframes are relative to each other and to the model's starting position. This means that if an object is moved in the scene (when, for example, a new version of the model is opened, or if movement tools are used in Autodesk Navisworks), the animation is done relative to the new starting location rather than the animation's original start position.

The translation, scaling and rotation operations are cumulative. This means if a particular object is in two animation sets at the same time, both sets of operations are carried out. So if both are translating across the X axis, for example, the object will move twice as far.

If there is no keyframe at the start of the timeline for an animation set, camera, or section plane set, then the very start of the timeline acts like a hidden keyframe. So, for example, if you have a keyframe a few seconds in, and the frame has the **Interpolate** option enabled, then over those first few seconds objects would interpolate between their default starting position and those defined in the first keyframe.

Edit Keyframes

You can edit captured keyframes for animation sets, cameras, and section plane sets.

To edit a keyframe

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 Right-click the desired keyframe in the timeline view, and select **Edit** on the shortcut menu.
- 3 Use the **Edit Key Frame** dialog box to adjust the animation.
- 4 Click **OK** to save your changes or **Cancel** to exit the dialog box.

Play Animation Scenes

Animations created in Autodesk Navisworks Manage 2012 can be played in all Autodesk Navisworks2012 products, including Freedom.

To play a scene in the Animator window

- 1 If the **Animation** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 From the **Scene Picker** drop-down list, select the scene you want to play in the **Animator** tree view.
- 3 Click **Play**  on the **Animator** toolbar.

To play a scene from the Animation tab

- 1 Click **Animation** tab ➤ **Playback** panel.
- 2 From the **Available Animations** drop-down list, select the scene you want to play.
- 3 Click **Play**  on the **Animation** toolbar on the **Playback** panel.

To adjust the scene playback

- 1 If the **Animator** window is not already open, click **Animation** tab
 ➤ **Create** panel ➤ **Animator**  .
- 2 Select the desired scene in the **Animator** tree view.

- 3 Use the **Loop**, **P.P.**, and **Infinite** check boxes to adjust the way the scene plays:
- If you want the scene to play back continuously, select the **Loop** check box. When the animation reaches the end, it will reset back to the start and run again.
 - If you want the scene to play in ping-pong mode, select the **P.P.** check box. When the animation reaches the end, it will run backward until it reaches the start. This will only happen once, unless you also select the **Loop** check box.
 - If you want the scene to play indefinitely (that is, until **Stop**  is clicked), select the **Infinite** check box. If this check box is unchecked, the scene will play until its end point is reached.

NOTE Selecting **Infinite** disables **Loop** and **P.P.**

- 4 If necessary, use the **Active**, **Loop**, and **P.P.** check boxes to adjust the playback of the individual scene components.

NOTE Only animations with the **Active** check box selected will play.

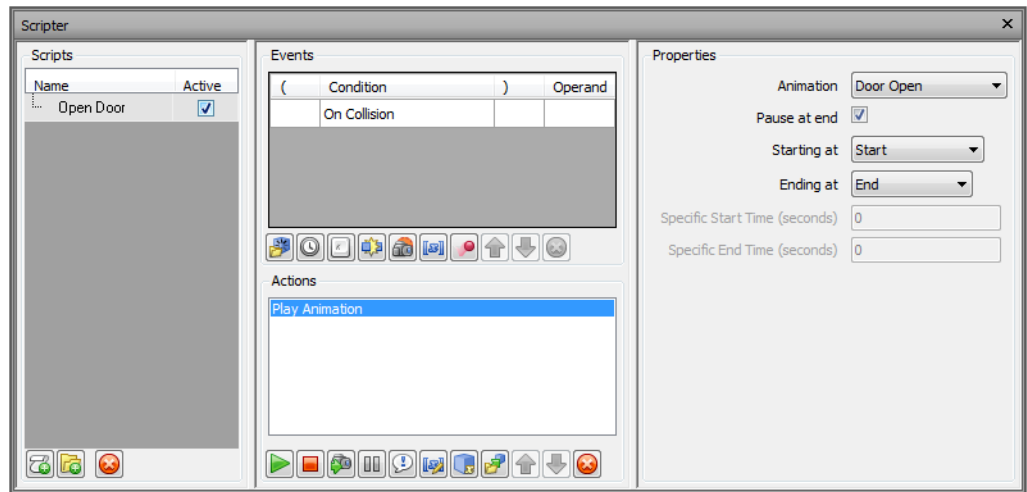
Add Interactivity

The **Scripter** window is a floating window that enables you to add interactivity to object animations in your model.

Ribbon: Animation tab ► Script panel ► **Scripter** 

 **Menu:** Classic user interface: Tools ► Scripter

 **Toolbar:** Classic user interface: Workspace ► Scripter 



The **Scripter** window contains the following components: the [tree view](#) (page 513), the [Events view](#) (page 515), the [Actions view](#) (page 517), and the [Properties view](#) (page 519).

NOTE If you can't use any of the controls in the **Scripter** window, it means that you are in interaction mode. To exit the mode, click **Animation** tab ➤ **Script** panel ➤ **Enable Scripts** .

Work with Animation Scripts

A script is a collection of actions that you want to happen when certain event conditions are met.

To add interactivity to your model, you need to create at least one animation script.

Each script can contain the following components:

- One or more events
- One or more actions

Your model can have as many scripts as you want, but only the active scripts are executed.

You can group the scripts into folders. This has no effect on their execution, except that the contents of a folder can be easily activated/deactivated to save time.

To add a script

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter** .
- 2 Right-click in the script view, and click **Add New Script** on the shortcut menu.
- 3 Click the default script name, and type in a new name.

TIP Use names that can help you to identify your scripts in future.

Pointing device: Click  at the bottom of the tree view.

To delete a script

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter** .
- 2 Right-click the script you want to delete in the tree view, and click **Delete Item** on the shortcut menu.

Pointing device: Click  at the bottom of the tree view.

NOTE If you accidentally delete an item, click **Undo**  on the **Quick Access** toolbar to restore it.

Shortcut menu: Right-click the script, and click **Delete Item**.

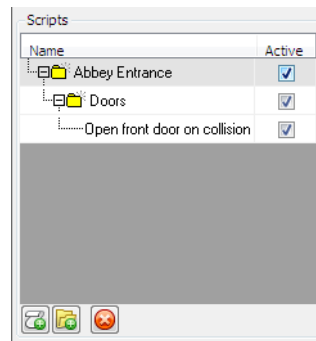
To organize scripts into folders

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter** .
- 2 Right-click in the tree view, and click **Add New Folder** on the shortcut menu.
- 3 Click the default folder name, and type in a new name.

TIP Use names that can help you to identify your scripts in future.

- 4 Select a script you want to add to your new folder. Hold down the left mouse button, and drag the mouse to the folder name. When the mouse pointer changes to an arrow, release the mouse button to drop the script into the folder.

Alternatively, select the script, and hold down the right mouse button. Drag the mouse to the folder name. When the mouse pointer changes to an arrow, release the right mouse button, and click **Move Here** on the shortcut menu.



Work with Events

An event is the occurrence of an incident or a situation, such as a mouse click, key press, or collision, which determines whether your script is run or not.

Your script can have more than one event in it. However, the way you combine all event conditions in the script becomes very important. That is you need to ensure the Boolean logic makes sense, the brackets are closed properly, and so on.

NOTE Until the combination of all event conditions in the script is satisfied, your script will not be executed.

Event Types

The following event types are available in Autodesk Navisworks:

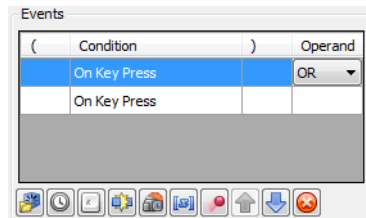
- **On Start** - event triggers a script as soon as scripting is enabled. If scripting is enabled when a file is loaded, then any start events in the file will be triggered immediately. This is useful for setting up the initial conditions

of your script, such as giving initial values to variables, or moving the camera to a defined start point.

- **On Timer** - event triggers a script at predefined time intervals.
- **On Key Press** - event triggers a script with a specific button on the keyboard.
- **On Collision** - event triggers a script when the camera collides with a specific object.
- **On Hotspot** - event triggers a script when the camera is within a specific range of a hotspot.
- **On Variable** - event triggers a script when a variable meets a predefined criterion.
- **On Animation** - event triggers a script when a specific animation starts or stops.

Event Conditions

Events can be combined with a simple Boolean logic. To create an event condition you can use a combination of brackets and AND/OR operators.



The brackets and logic operators can be added by right-clicking an event, and selecting the option from the shortcut menu. Alternatively, you can click in the corresponding field in the **Events** view, and use the drop-down list to select the desired option.

NOTE If you have more than one event in your script, by default the AND operator will be used. This means the script only runs if all events have occurred.

To add an event

- 1 If the **Scripter** window is not already open, click **Animation** tab
 - **Script** panel ➤ **Scripter** .
- 2 Select the desired script in the tree view.

- 3 Click the desired event icon at the bottom of the **Events** view. For example, click  to create an **On Start** event.
- 4 Review the event properties in the **Properties** view, and [adjust them as necessary](#) (page 519).

To test event logic

- 1 If the **Scripter** window is not already open, click **Animation** tab
 > **Script** panel > **Scripter** .
- 2 Select the desired script in the tree view.
- 3 Right-click the **Events** view, and click **Test Logic** on the shortcut menu.
Autodesk Navisworks checks the event conditions in your script, and reports back any detected errors.

To delete an event

- 1 If the **Scripter** window is not already open, click **Animation** tab
 > **Script** panel > **Scripter** .
- 2 Select the desired script in the tree view.
- 3 Right-click the event you want to delete in the **Events** view, and click **Delete Event**.

Work with Actions

An action is an activity, such as playing or stopping an animation, showing a viewpoint and so on, which is carried out when a script is triggered by an event or a combination of events.

Your script can have more than one action in it. Actions are executed one after another, so it is important to get the action sequence right.

NOTE Autodesk Navisworks does not wait for the current action to be completed before moving on to the next action.

Action Types

The following action types are available in Autodesk Navisworks:

- **Play Animation** - action specifies which animation to play back when a script is triggered.
- **Stop Animation** - action specifies which currently playing animation to stop when a script is triggered.
- **Show Viewpoint** - action specifies which viewpoint to use when a script is triggered.
- **Pause** - enables you to stop the script for a specified amount of time before the next action is run.
- **Send Message** - action writes a message in a text file when a script is triggered.
- **Set Variable** - action assigns, increases or decreases a variable value when a script is triggered.
- **Store Property** - action stores an object property in a variable when a script is triggered.
This can be useful if you need to trigger events based on embedded object properties or live data in a linked database.
- **Load Model** - action opens a specified file when a script is triggered.
You may find it useful if you want to present a selection of animated scenes contained in a range of different model files.

To add an action

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter** .
- 2 Select the desired script in the tree view.
- 3 Click the desired action icon at the bottom of the **Actions** view. For example, click  to add a **Play Animation** action.
- 4 Review the action properties in the **Properties** view, and [adjust them as necessary](#) (page 522).

To test an action

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter** .

- 2 Select the desired script in the tree view.
- 3 Right-click the **Actions** view, and click **Test Action** on the shortcut menu.
Autodesk Navisworks executes the selected action.

To delete an action

- 1 If the **Scripter** window is not already open, click **Animation** tab
 ➤ **Script** panel ➤ **Scripter**  .
- 2 Select the desired script in the tree view.
- 3 Right-click the action you want to delete in the **Actions** view, and click **Delete Action**.

Enable Scripting

To enable animation scripts in your file, you need to click **Animation** tab

➤ **Script** panel ➤ **Enable Scripts**  .

You can now interact with your model.

NOTE When the scripts are enabled, you can't create or edit scripts in the **Scripter** window. To disable scripting, click **Animation** tab ➤ **Script** panel ➤ **Enable Scripts**  again.

Create Photorealistic Visualizations

13

You can use the **Presenter** to apply texture materials, lighting, Rich Photorealistic Content (RPC), and background effects to your model.

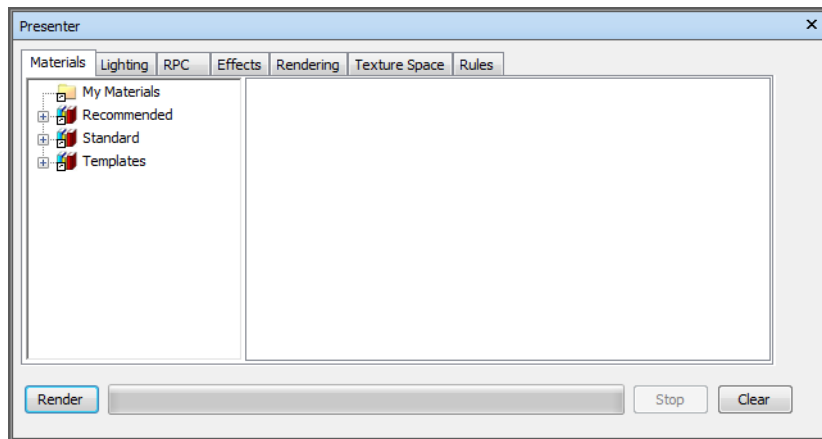
NOTE Presenter materials are only available when using Presenter graphics system.

Overview of the Presenter Tool

Presenter Window

The **Presenter** dockable window is used to set up materials and lights in your scene and render it with more realism and effects. You can also use it to edit predefined materials and apply them to items in the scene, add lights to the scene, and set up rules for applying the materials to other files in the same project set up with the same parameters. You can define and apply your materials and lights to a model and save the settings into an NWF file so that as the model is updated, the materials and lights remain the same.

Materials can also be brought through from CAD applications via the 3DS, DWG, and DGN file formats, or by exporting from 3D Studio Viz or Max. See [Work with Files](#) (page 163) for more details on this.



The **Presenter** window contains the following tabs:

- **Materials.** Includes a variety of materials, which can be selected and applied to individual or groups of model items. You can also use this tab to create new materials, or customize existing materials.
- **Lighting.** Includes a variety of lighting options, which can be selected and applied to the model. You can also customize lighting options, if you want.
- **RPC.** Contains Rich Photorealistic Content (or RPC), which you can add from various sources, including websites. RPCs can include images of people, trees, cars and so on.
- **Effects.** Includes a variety of backgrounds and environments, which can be selected and applied to the model scene. Some existing backgrounds can be customized, or new backgrounds can be created. You can also add backgrounds and environments from other sources, such as websites.
- **Rendering.** Includes a variety of rendering styles, which can be selected and applied to the model. A render style affects the way a scene is rendered. You can also use this tab to create new render styles, or customize existing render styles.
- **Texture Space.** Defines the way in which a texture is applied to a model item, for example, applying a cylindrical texture space to a pipe will provide a more natural effect.
- **Rules.** Applies materials to models according to user-defined criteria. Rules can be used, for example, to speed up the application of materials to groups of items.

The **Materials**, **Lighting**, **Effects**, and **Rendering** tabs are divided into two panes. The left pane contains the [archives](#) (page 552) and the right pane contains the palette, which defines what materials, lighting, effects, and render styles are used in the scene. Archives are shown in a tree structure and are defined in the LightWorks Archive (.lwa) format.

Model Appearance

While **Presenter** can be used for photo-realistic renderings, it can also be used for OpenGL interactive rendering. Once you've set up the scene with **Presenter**, you can view the materials and lights in Autodesk Navisworks in real time.

For more information, see [Control Model Appearance](#) (page 317).

To toggle the Presenter window

■ Choose **Home** tab ► **Tools** panel ► **Presenter** .

Ribbon: **Home** tab ► **Tools** panel ► **Presenter** 

 **Menu:** Classic user interface: **Tools** ► **Presenter**

Use the Presenter Archives

There are three predefined archives that are installed with **Presenter**:

- The **Recommended** archives contain materials, lighting, effects, and rendering styles that are recommended for most users. These include materials, lights, and effects that can be seen during interactive navigation in Autodesk Navisworks and can be fully rendered with OpenGL. Of course, they will look better when rendered photorealistically.
- Additional materials, light studios, effects, and render styles are available in the **Standard** archives. These include materials that cannot be fully reproduced using OpenGL and therefore will not be seen properly in interactive mode, or until a full photorealistic render is done.
- Any materials, lights, effects, and render styles from any archive can be used as a template starter for your own definitions, but the **Templates** archives contain instances of each type of material, light, effect, and render style, giving you quicker access.

TIP Additional LWA archives can be downloaded from [Lightworks-User](#), and added to the **Materials**, **Lighting**, **Effects**, and **Rendering** tabs.

Although materials, lights, effects, and render styles cannot be edited in archives, once dragged into the scene's palette, they can be edited and saved with the scene in an NWF file, or published as an NWD file.

You can save your own edits to materials into an NWP file format. See [Organize and Manage Materials](#) (page 559) for more information on how to do this.

The User Archives

The **user** archives allow you to save your own edited materials, lights, effects, and render styles for use in other scenes.

The **User** archives are accessible from each of the **Materials**, **Lighting**, **Effects**, and **Rendering** tabs. On each of the tabs, the archives are named **My Materials**, **My Lighting**, **My Effects**, and **My Render Styles**, respectively.

To manage user archives

- To save a material, light, effect, or render style to a user archive for use in other scenes, in the right pane, drag that item from the scene's palette onto the respective user archive.
- To create a new subfolder in a user archive, right-click the archive, and click **New Directory** on the shortcut menu. You can rename this new folder by right-clicking it, and clicking **Rename** on the shortcut menu. You can add as many nested subfolders as you want.
- To save a user archive to disk, right-click it, and click **Save Archives** on the shortcut menu. This saves any modified archives.

NOTE You will also be prompted to save any modified archives when you close Autodesk Navisworks.

- To remove a material, light, effect, or render style from the respective user archive, right-click the item to be removed, and click **Delete** on the shortcut menu.
- To remove a subfolder you've created from the respective user archive, right-click the folder to be removed, and click **Delete** on the shortcut menu.

NOTE You cannot delete any default user archive folders.

To add archives

- To download an archive from the [Lightworks-User](#) website, in the left pane, right-click any archive, and click **Download Archive** on the shortcut menu. Then follow the instructions given on the site.
- To import a downloaded archive into **Presenter**, in the left pane, right-click any archive, and click **Import Archive** on the shortcut menu. Use the standard **Open** dialog box to select the LWA file to import.

To delete an added archive

- 1 Right-click the archive, and click **Delete Archive** on the shortcut menu.
- 2 Click **Yes**.

Photo-Realistic Scene Rendering

You can render directly in the **Scene View** by clicking **Render** at the bottom of the **Presenter** window at any time.

You can export rendered scenes as images, and use them in presentations, on websites, in print, and so on. You can also export animated AVI presentations and instructional movies, in which the animated objects move in photo-realistically rendered scenes.

NOTE The larger the image size of the exported animation, the higher the resolution. However, this significantly increases render times. If you are creating high-resolution images, it is best to use a separate machine to render.

Once you have set up and rendered a scene, you can additionally create animation in that scene. The rendering that you have set up will be applied to each frame of the animation.

To set up and render a scene

- 1 Choose **Home** tab ► **Tools** panel ► **Presenter**  to open the **Presenter** window.
- 2 Set up the scene.
 - Use the **Materials** tab to drag and drop materials onto items in the model.

You can use the predefined materials, or create your own from the templates in the **Materials** tab (see [Use Presenter Materials](#) (page 556) for more details).

Or

Use the **Rules** tab to set up rules which define project-wide material application (see [Use Presenter Rules](#) (page 600) for more details).

- Use the **Texture Space** tab to more accurately map materials onto items in the scene (see [Use Presenter Texture Space](#) (page 598) for more details on this).
 - Use the **Lighting** tab to set up additional lighting (see [Use Presenter Lights](#) (page 568) for more details).
 - Use the **Effects** tab to add background and foreground effects to the scene (see [Use Presenter Rendering Effects](#) (page 587) for more details).
 - Use the **Rendering** tab to select a rendering style for the render (see [Use Presenter Rendering Styles](#) (page 594) for more details).
- 3 At any point, you can click **Render** to start the rendering process in the **Scene View**. The rendering process can be stopped at any point by clicking **Stop**.
 - 4 Click **Clear** to clear the render in the **Scene View**, and return to an OpenGL interactive view.

To print a rendered image

- 1 Apply materials and lighting effects to the required scene, and click **Render**.
- 2 When the scene is rendered, click the **Output** tab ► **Visuals** panel ► **Rendered Image**.
- 3 In the **Export Rendered Image** dialog box, select **Printer** from the **Type** drop-down list.
The **Browse** options are grayed out.
- 4 Click **OK**.
- 5 In the **Print** dialog box, select the desired printer, enter the print settings, and click **OK**.

Ribbon: **Output** tab ► **Visuals** panel ► **Rendered Image** 
 **Menu:** Application button ► **Export** ► **Images & Animations** ► **Rendered Image**

To save a rendered image

- 1 Apply materials and lighting effects to the required scene, and click **Render**.

- 2 When the scene is rendered, click the **Output** tab ➤ **Visuals** panel
➤ **Rendered Image** .
- 3 In the **Export Rendered Image** dialog box, select the desired file type from the **Type** drop-down list. For more information, see [Export Rendered Image Dialog Box](#) (page 753)
- 4 Browse to a location and enter the name of the file you want to render to.
- 5 Set the **Size** of the rendered file.

NOTE If you choose **Use View** as the **Size** of the image file, then Autodesk Navisworks will save any existing render in the **Scene View**, without having to render again from scratch.

- 6 Click **OK**.

Ribbon: **Output** tab ➤ **Visuals** panel ➤ **Rendered Image** 

Menu: Application button ➤ **Export** ➤ **Images & Animations** ➤ **Rendered Image**

To export rendered animations

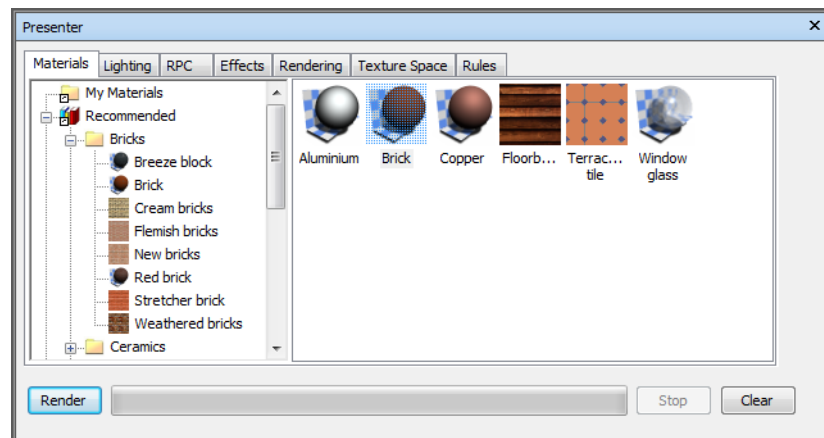
- 1 Apply materials and lighting effects to the required scene, and, in the **Presenter** window, click **Render**.
 - 2 When the scene is rendered, click the **Output** tab ➤ **Visuals** panel
➤ **Animation** .
- The **Animation Export** dialog box opens.
- 3 From the **Source** drop-down list, select the animation type you want to export.
 - To export an object animation, select **Current Animator Scene**.
 - To export a TimeLiner sequence, select **TimeLiner Simulation**.
 - To export a viewpoint animation, select **Current Animation**.
 - 4 Set the remaining options in the **Animation Export** dialog box, and click **OK**. For more information, see [Animation Export Dialog Box](#) (page 737).
 - 5 In the **Save As** dialog box, enter a new file name and location, if you want to change from those suggested.
 - 6 Click **Save**.

Ribbon: Output tab ► Visuals panel ► Animation 
Menu: Application button ► Export ► Images & Animations ► Animation

Use Presenter Materials

Materials Tab

Like the [Lighting](#) (page 568), [Effects](#) (page 587), and [Rendering](#) (page 594) tabs, the **Materials** tab is divided into two panes. The left pane describes the predefined [archives](#) (page 552) of materials that are installed and the right pane shows the current palette of materials that have been defined and are being used in the scene. The palette also shows a small thumbnail of each material as it will appear when rendered.



Apply and Remove Presenter Materials

Materials can be applied to items in the scene by dragging and dropping the material onto:

- An item in the **Scene View**
- An item in the **Selection Tree**
- A selection or search set

If you drag the material from an archive, it will appear in the palette where it can be edited and saved with the scene, if necessary.

Presenter uses Autodesk Navisworks selection resolution to decide which items to apply the material to when dragging from an archive or palette into the main view. When hovering over any item in the main view, the proposed selection will change into the selection color (blue by default). When you drop the material onto the current selection, it will be applied to all the items selected. If you drop the material onto an item that is not currently selected, it will be applied to just that item. See [Set Selection Resolution](#) (page 345) for more information on selection resolution.

You can also apply materials to items by selecting the items in the **Selection Tree** or **Scene View**, right-clicking the material in the palette, and clicking **Apply to Selected Items**.

Rules can also be used to apply materials to items automatically based on their layer or color or selection set names, for example. See [Use Presenter Rules](#) (page 600) for more information on this.

You can remove materials assigned to geometry items, either from the **Presenter** window, or directly in the **Scene View** or **Selection Tree**.

Inheritance

Layers can have colors, just as geometry can. If a layer has a material, all its children in the **Selection Tree** inherit this material, until one of the children is assigned its own material, at which point, all *its* children in the **Selection Tree** inherit this material, and so on.

You can drag and drop materials onto layers. Only the layer picks up the material, and although its children inherit the material, they do not have it explicitly assigned to them.

Therefore, right-clicking such a child will not allow you to remove the material because one was not explicitly assigned in the first place.

However, if you use a rule to assign a material to a certain color, then all objects in the scene will get this material explicitly assigned to them, including parent layers and child objects. If, with a selection resolution of something like **Geometry** (which is more specific than a resolution of **Layer**), you right-click a child object, and click **Remove Materials** on the shortcut menu, then the material will be removed from the child object, but not from the parent layer and there won't be any apparent difference.

To remove the material, you will, therefore, have to remove it from the parent object; in the above situation this would be the layer.

To apply materials to model geometry

- 1 Select the geometry items directly in the **Scene View** or in the **Selection Tree**.
- 2 Open the **Presenter** window, and click the **Materials** tab.
- 3 Select your material from an archive or palette, right-click this material, and click **Apply to Selected Items** on the shortcut menu to assign the material to the current geometry selection.

NOTE Selected material will only be applied to the selected geometry item and not to every instance of the item. To assign the material to all instances of the selected geometry in the scene, click **Apply to All Instances of Selection** on the shortcut menu.

TIP You can drag and drop a material from an archive or palette onto items in the **Selection Tree** or **Scene View** to assign those materials to the items. The selection resolution determines which items will receive the materials.

To use Scene View or the Selection Tree to remove materials from model geometry

- Right-click the item in the **Scene View**, or in the **Selection Tree**, and click **Presenter > Remove Material** on the shortcut menu.

NOTE **Remove Material** will only be available if the right-clicked item has a material assigned to it at that selection resolution in the **Selection Tree**.

To use the Presenter window to remove materials from model geometry

- 1 Open the **Presenter** window, and click the **Materials** tab.
- 2 In the material palette, right-click the material that you want to remove from items in the scene, and click **Remove From All Items** on the shortcut menu.

NOTE Deleting the material from the palette will automatically remove that material from any items in the **Scene View** which it was applied to.

Organize and Manage Materials

Custom Folders

You can organize materials into custom folders for easy reference and management. By doing this, you are, effectively, customizing a user archive. For more information, see [Use the Presenter Archives](#) (page 551).

The Material Palette

On the **Materials** tab, the right pane, or material palette, is where you edit and manage your materials for your scene. Materials are taken from the archives into the palette where they are edited. You can then save the palette into a Autodesk Navisworks palette file (NWP) for use in other scenes too.

To add a custom folder

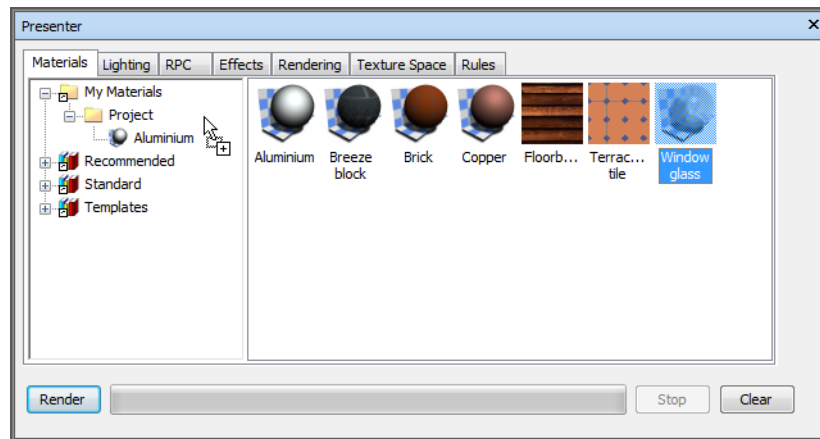
- 1 Open the **Presenter** window, and click the **Materials** tab.
- 2 Right-click the **My Materials** folder in the left pane, and click **New Directory** on the shortcut menu.
- 3 Expand the **My Materials** folder, right-click the new folder, and click **Rename** on the shortcut menu.
- 4 Type in the new name, for example “GGK Project”.

To delete a custom folder

- 1 In the **Presenter** window, **Materials** tab, expand the **My Materials** folder in the left pane, and right-click the folder you want to delete.
- 2 Click **Delete** on the shortcut menu.

To copy materials into a custom folder

- 1 In the **Presenter** window, **Materials** tab, click a material in the right pane, and drag it to the desired location under the **My Materials** folder in the left pane until a mouse pointer displays a small plus symbol.
- 2 Release the left mouse button to drop the material into the folder.



NOTE You can also copy materials between the folders in the left pane by either dragging and dropping them, or using the **Copy** and **Paste** options on the shortcut menu.

To manage palette materials

- 1 Right-click a material in the right pane of the **Materials** tab (the palette).
 - Click **Copy** to copy the material to the clipboard. Right-click an empty space in the palette and click **Paste** to paste a copy of the material with the same name suffixed with the next number in the list. This process is useful if you want to test small tweaks to a material.
 - Click **Delete** on the shortcut menu to delete the material from the palette. This will also remove the material from all items in the scene.
 - Click **Regenerate Image** to regenerate the thumbnail of the material in the palette with the current attributes.
 - Click **Rename** to rename the material. You can also select the material and press **F2** to rename it.
 - Click **Edit**, or double-click a material to open the **Material Editor** dialog box, allowing you to edit its parameters. See [Edit Presenter Materials](#) (page 561) for more information.
 - Depending on whether items are selected in the scene and whether the material has been assigned to any items, there will also be a couple of **Apply** and **Remove** items on the shortcut menu. See [Apply and Remove Presenter Materials](#) (page 556) for more details.

- Click **Select All Instances** to select the items in the scene which have this particular material assigned to them.
- Click **Load Palette** to load a previously saved palette of materials into the current scene. This will delete any materials currently in the palette. The standard **File Open** dialog box opens, allowing you to browse to an NWP file.
- Click **Append Palette** to load a palette from an NWP file, while keeping all the existing materials in the current palette. Any materials that are duplicated will be renamed with the NWP file as an extension.
- Click **Merge Palette** to merge a palette from an NWP file into the current scene. This is like appending, but instead of adding and renaming any duplicate materials, merging will overwrite existing materials of the same name.
- Click **Save Palette As** to save your current palette of materials into a Autodesk Navisworks Palette NWP file. If you save the current scene using the usual **Save** method into an NWF or NWD file, the palette will be saved too, but the independent NWP file is useful if you want to transfer materials you've made in one scene into other scenes.

NOTE If you publish an NWD file using **Output** tab ➤ **Publish** panel ➤ **NWD**, a **_Presenter_Maps** folder will be created along with the NWD file. This folder will contain any materials that are not contained in **Presenter Runtime**, which is used by both Autodesk Navisworks Manage 2012 and Autodesk Navisworks Freedom 2012, to view materials.

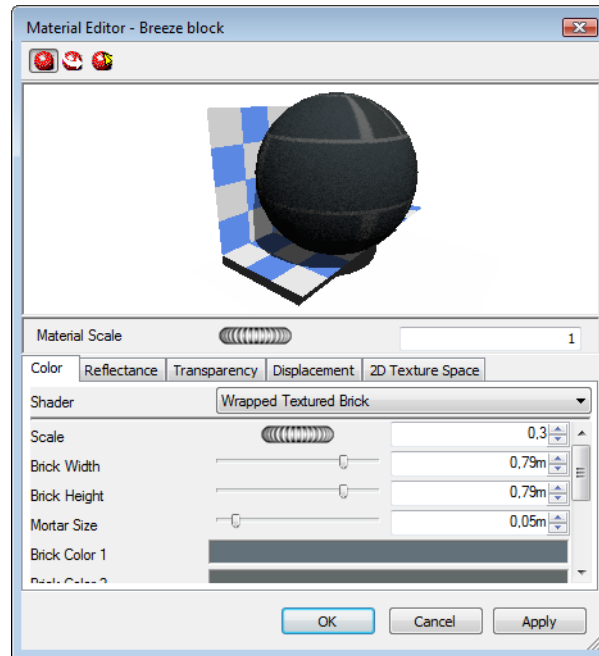
- Click **Clear Palette** to delete all the materials from the palette and also from all items in the scene.

Edit Presenter Materials

Installed archive materials cannot be edited while they are in the archives, but you can edit materials in the scene's palette. Edited materials will be saved with the Autodesk Navisworks model in an NWD or NWF file, or in an NWP palette file, or they can be added to your user archive, called **My Materials**.

To edit a material, double-click it in the palette, or right-click it and click **Edit** on the shortcut menu. The **Material Editor** dialog box opens, which will vary for different types of material. You can't add or remove parameters on a material. You can merely edit those existing. So it is important to use the right type of material template for the material you want to edit. The dialog box

for the **Breeze Block** procedural texture is shown below and this will be used as an example of how to edit a material.



NOTE If the **Presenterprofile** in the **Options Editor** (see [Presenter Page](#) (page 824) for more information) is set to **Advanced**, there are more tabs and parameters to edit in this dialog box. In particular, there are **Reflectance**, **Transparency**, **Displacement**, and **2D Texture Space** tabs and at the top of each tab is a **Shader** type which allows you to completely change the type of material and all other parameters.

To edit the Breeze Block texture

- 1 Open the **Presenter** window, and click the **Materials** tab.
- 2 Expand the **Recommended\Bricks** material archive, and double-click **Breeze Block** in the palette to open the **Material Editor**.
- 3 The three buttons at the top determine what sort of material preview you get:
 - Click **Standard Preview**  to get a software generated photorealistic preview of the material on the standard ball against

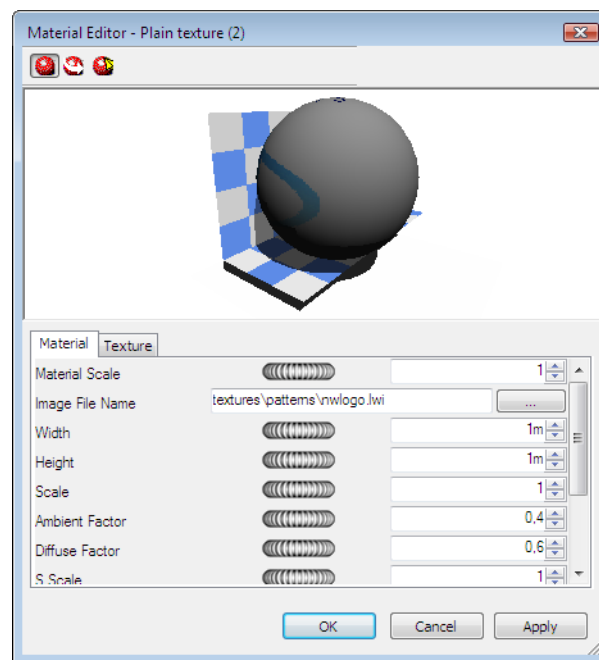
checkered background. Though not interactive, the preview will show how the material will look when rendered photorealistically.

- Click **Active Preview**  to get an OpenGL interactive preview of the material on the standard ball against checkered background. This is updated interactively while you change the parameters. The preview will resemble the quality of material shown in Autodesk Navisworks during navigation, but the quality will not be as high as the photorealistic render.
 - Click **Main Window Preview**  to close the preview window in the **Material Editor** and instead preview the material on the item in the **Scene View**. This is updated interactively while you change the parameters. The preview will be represent the material shown in Autodesk Navisworks during navigation, but the quality will not be as high as the photorealistic render.
- 4 For a simple material, there is only a single **Material** tab on the **Material Editor**, whereas for a texture material, whether procedural (generated from an algorithm) or bitmap (generated from an image), there is an extra tab called **Texture**.
- The **Material** tab contains simple parameters that affect the material's color, scale, shininess and so on. In the case of the **Breeze Block**, there are parameters for the overall scale of the material, as well as a block's width and height, the block's color and mortar color, its roughness and reflectivity. For bitmap textures, you need to define the location of the image in the **Image File Name** box. On a glassy material, other factors would affect the transparency and refraction properties of the glass. Some of these factors will not be apparent in the interactive OpenGL window and will have to be rendered with the **Render** to be seen.
 - The **Texture** tab contains parameters that specifically affect a texture material's texture mapping properties, such as its rotation, offset (origin) and S- and T- (sometimes called U- and V-) scales. These parameter values are applied in relation to an origin point (see [Advanced Materials](#) (page 565) for more information). There are S- and T- Reflect check boxes, which will show the reflection of the image in either (or both) of these axis. Finally there is an **Offset Center** check box, that repositions the origin to the center of the image (again, see [Advanced Materials](#) (page 565) for more information). When using the **Main Window Preview**, texture changes can be made instantly allowing interactive positioning of materials on an object.

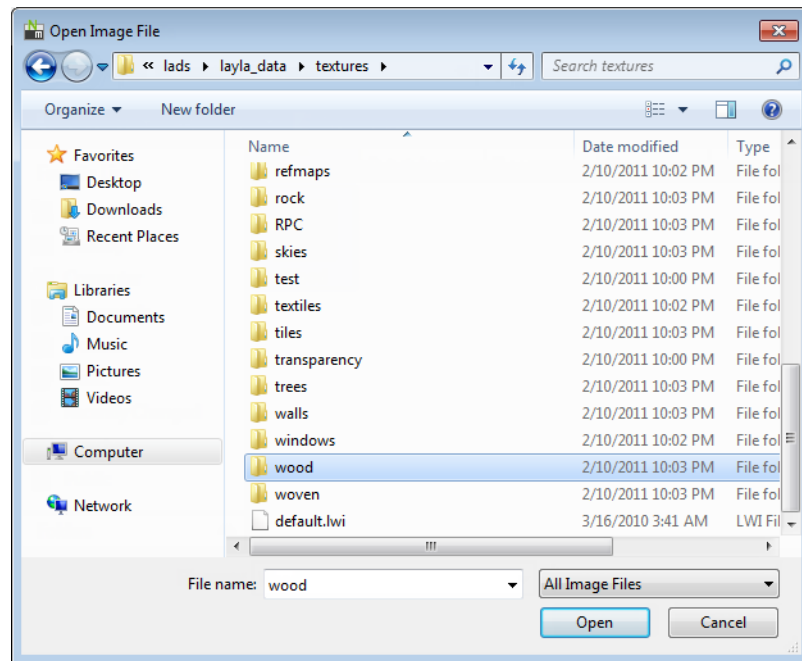
- 5 At any time, click **Apply** to apply the parameter edits to the material in the scene.
- 6 Click **OK** to keep the changes made or **Cancel** to discard any changes made (since the last time you clicked **Apply** at least).

To create a texture using your own image

- 1 Expand the **Templates** material archive, and double-click the **Plain Texture** material to add the material to the scene's palette, and to open the **Material Editor**.



- 2 On the **Material** tab (or **Color** tab, if in **Developer** profile) click the **Browse** button (...) next to the **Image File Name** text box. The **Open Image File** dialog box opens.



- 3 Browse to and select your image file, and click **Open**.
- 4 You may then need to adjust some of the texture parameters of the new material, for example its scale, rotation, offset or reflection (if it's back to front). These may all be edited in the **Texture** tab. See [Edit Presenter Materials](#) (page 561) for more information.

Advanced Materials

Internally, a material is defined by four shaders from different classes: **Color**, **Transparency**, **Reflectance** and **Displacement**. Each class of shader controls a different aspect of a material's behavior. There are many types of shader in each class, each type being defined by its own set of parameters.

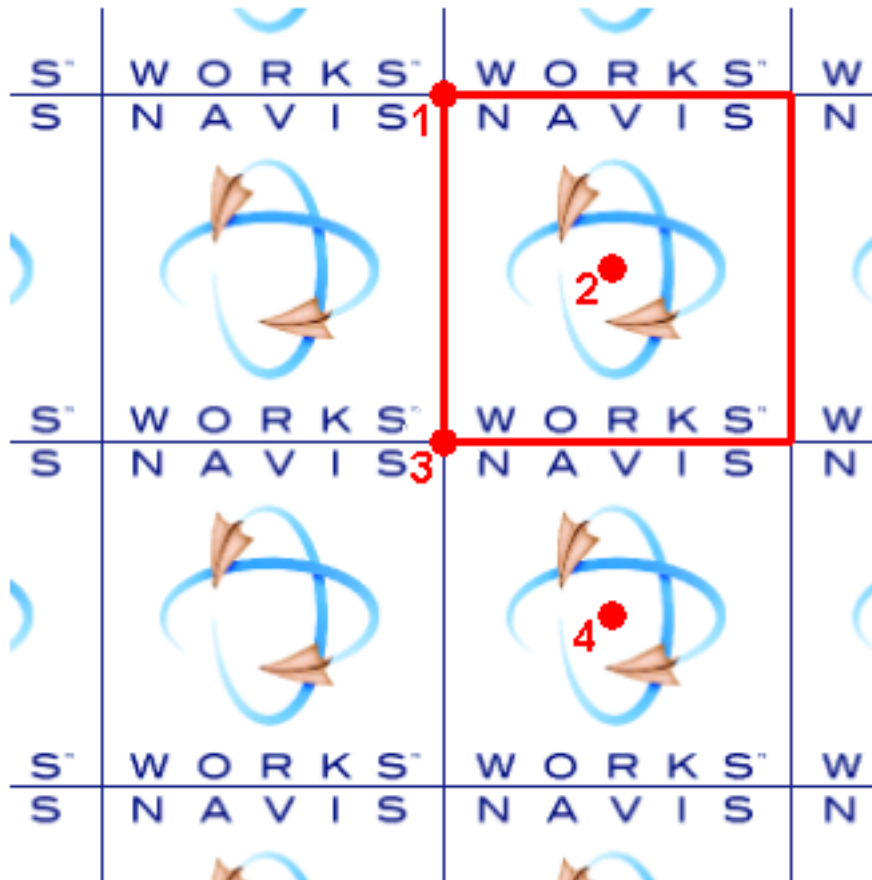
- A **Color** shader is used to define the color of a surface at any point in space. It may be as simple as a plain color which specifies all parts of the surface to have a uniform color, or it may define complex surface patterns such as marble or wood. Every material *must* have a color shader.
- The behavior of a surface in the presence light is represented by a **reflectance** shader which defines how much light is reflected by the

surface towards the viewer. Shaders of this class may be thought of as defining a surface's "finish", and are used to model properties such as matte, metal, and plastic.

- A **transparency** shader is used to define how transparent or opaque a surface is, and thus how much light is able to pass through it. Transparency shaders range from a simple uniform transparency to more complex regular or irregular eroded patterns that would be more difficult to represent using modelling techniques. A material without a transparency shader is completely opaque.
- Small surface perturbations can be supported by means of **displacement** shaders. Typically, a displacement shader will give an otherwise smooth surface an irregular or indented appearance. Displacement shaders are used to represent features that would be difficult, impossible, or inefficient if conventional modelling techniques were used. For example, rough metal castings and the regular indentations produced by pressed sheet metal can be simulated.

Normally, the **Material Editor** displays a selection of the most important parameters from all shaders within the **Materials** tab. If the user profile is set to **Developer** on the **Interface** node in the **Options Editor** (see [Presenter Page](#) (page 824)), then all four shaders can be edited and changed individually.

Some shaders are described as "wrapped". These define a flat, two dimensional material, like wallpaper. Wrapped materials need a **texture space** shader to define how they should be applied to (wrapped around) a three dimensional object. Materials that include a wrapped shader can also include a texture space shader. A special type of texture space shader, called a **layout** shader, can be used to transform (rotate, stretch, offset) the two dimensional material before it is applied to the three dimensional object. Transforms are based around an origin point, which by default, is the top left corner of the image (refer to the diagram below, where the image is inscribed in the red square, which is then repeated. The default origin is **Point 1**). Selecting the **Offset Center** check box will reposition the origin to the center of the image (**Point 2**). Finally, in **Developer** profile, you can edit the **Decal Mode**, choosing from either **Default** or **Normalized**. Selecting **Normalized** will move the origin to the lower-left corner of the image (**Point 3**, with the **Offset Center** option cleared). With both **Normalized** and **Offset Center** selected, the origin will be repositioned in the center of the repeated image, directly below (**Point 4**).



In the **Presenter** window, materials that include a wrapped shader also have a layout **texture space** shader associated with them. Normal texture space shaders are associated with objects.

NOTE A complete reference manual for all types of rendering styles is included with the Autodesk Navisworks API, (see \API\COM\documentation\shaders.chm). The Autodesk Navisworks API is included with Autodesk Navisworks Manage 2012; it's an optional feature in the installer, and is installed by default into the API subfolder in the Autodesk Navisworks installation folder.

Use Presenter Lights

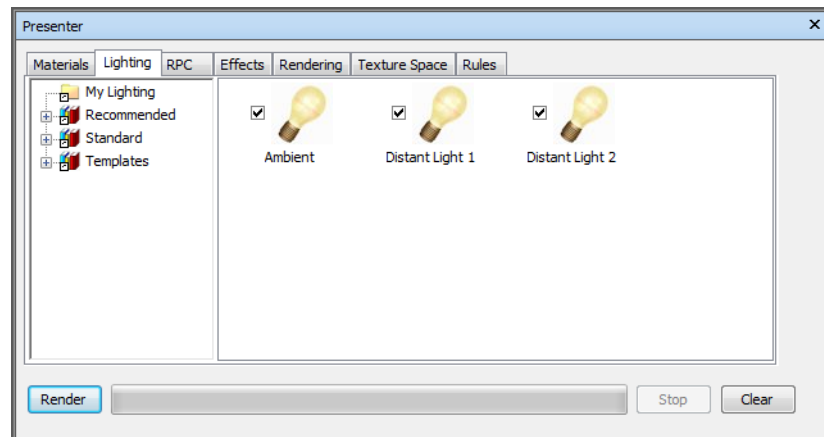
Lighting Tab

Like the [Materials](#) (page 556), [Effects](#) (page 587) and [Rendering](#) (page 594) tabs, the **Lighting** tab is divided into two panes, with the archives on the left and the palette on the right. The archive contains individual lights, as well as light studios. A light studio is a combination of lights that work well together. The palette contains all the lights that are active in the scene.

To apply a light to the scene, drag it from an archive into the palette, at which point you can edit its parameters if required. The light is added to those already in the scene.

To apply a light studio to the scene, drag it from an archive into the palette. All the lights in the light studio *replace* those already in the scene. Light studios are applied to the scene intelligently. The light studio is oriented and scaled to match the scene to which it is being applied. You can also expand a light studio in the archive and drag the lights into the palette individually. If you do this, the lights are *not* oriented or scaled to match the scene.

Each light in the palette has a check box to turn the light on or off in the scene.



Add and Position Lights

Lights and light studios can be taken directly from the archives and applied to the model by dropping them into the palette. These can then be repositioned as needed.

The **Recommended** archive contains five lights (**Ambient**, **Distant**, **Eye**, **Point**, and **Spot**), a **Standard Light Studio**, an **Environment Light Studio**, an **Environment** folder containing two Environment lights utilizing HDRI-based light sources (see [Image-based Lighting](#) (page 580) for more information), and an **Exterior** folder with different city locations around the world, each containing three light studios (**Clear Sky**, **Overcast Sky** and **Sun Study**).

If you are creating an external render of a building, for example, then you may find that one of the **Environment** light studios can give a very realistic effect, using image-based lighting to light the model. **Exterior** light studios may also give you the effect you require. These do use physically accurate lights, however, which generally take longer to render the scene.

Alternatively, you may prefer to use the **Standard Light Studio** as a starting point and build up your lighting from there, adding combinations of the basic recommended lights to create the desired effect.

The **Standard** archive contains a **Default Eye Light** studio (which is effectively rendering with a head light); an **Environmental** folder; a folder of **Exterior** light studios, which predominantly consists of light studios that use a number of lights to replicate the effect of a **Sky** light. Not using physically accurate lights means, you don't have to turn on **Auto Exposure** (see [Auto Exposure](#) (page 598)), which can negatively impact on the basic recommended light settings; a folder of **Interior** light studios for use in internal scenes; a folder of **Object** light studios which are best suited to lighting smaller models, such as a vehicle or piece of machinery, for example; and a folder of **Projector** light studios, which can be used to project an image onto an object in the scene; and a **Simple Sky** folder.

The **Templates** archive contains all of the basic light shaders that are available. These can then be edited (as can all lights) to create the exact lighting you require (see [Edit Lights](#) (page 573) for more information).

To add lights to the model

- 1 Open the **Presenter** window, and click the **Lighting** tab.
- 2 In the left pane, choose the light you want to add to the scene.

- 3 Drag the light and drop it into the palette (right pane of the **Lighting** tab). This will automatically be added to the scene.

NOTE If you drag a light studio into the palette, this will replace any existing lights with those that make up the light studio.

As a general guide, the more lights there are in a scene, the longer it will take to render it photorealistically. For external rendered scenes, you may consider using the **Standard Light Studio** from the **Recommended** archive as a starting point, then strategically add a couple of **Point** and **Spot** lights around the scene. **Point** lights are good to light up a dark area of the scene, while **Spot** lights can add an element of drama and enhance realism.

To position or reposition lights in the model

- 1 Open the **Presenter** window, and click the **Lighting** tab.
- 2 In the palette (right pane of the **Lighting** tab), right-click the light you want to reposition, and click **Edit** on the shortcut menu.
- 3 Use the **Light Editor** to position the light as needed.

Point, **Distant**, **Spot** and **Projector** lights have a **Location** parameter. **Distant** and **Spot** lights additionally have a **To** parameter. You can type in X-, Y-, and Z- coordinates for these, or alternatively use the **Pick** button to interactively pick a point in the scene where the light and/or target is located. The light is represented by a 3D wireframe sun icon in the scene and the target by a wireframe sphere. The currently selected light is drawn in the selection color (see [Selection Page](#) (page 768) for more information).

NOTE Autodesk Navisworks does not allow you to pick a point in empty space so you must pick a point on the model.

- 4 Lights can be positioned interactively. The 3D wireframe sun icon, representing the light, has six bars extending out along the X- axis, Y- axis and Z- axis. Hover the mouse pointer over one of the bars. The mouse pointer will change to a hand and the bars will extend further along that axis. Hold the left mouse button down to hold on to the light and move it in either direction, along the extended bar. Release the left mouse button to release the light in its new position. This can be performed for all three axis.
- 5 Lights can also be positioned in the current location of the camera, which can be anywhere in the scene. Navigate to the location where you want the light to be positioned. Right-click the light in the palette (right pane

of the **Lighting** tab), and click **Position as Camera** on the shortcut menu.

NOTE Not only will this position the light in the same location as the camera, if the light has a **To** parameter, this will also be set to the focal (or **Look At**) point of the camera.

Organize and Manage Lights

Manage Folders

You can organize lights into custom folders for easy reference and management. By doing this, you are, effectively, customizing a user archive. For more information, see [Use the Presenter Archives](#) (page 551).

The Lighting Palette

On the **Lighting** tab, the right pane, or lighting palette, is where you edit and manage your lights for your scene. Lights are taken from the archives into the palette where they are edited.

To add a custom folder

- 1 Open the **Presenter** window, and click the **Lighting** tab.
- 2 Right-click the **My Lighting** folder in the left pane, and click **New Directory** on the shortcut menu.
- 3 Expand the **My Lighting** folder, right-click the new folder, and click **Rename** on the shortcut menu.
- 4 Type in the new name, for example “GGK Project”.

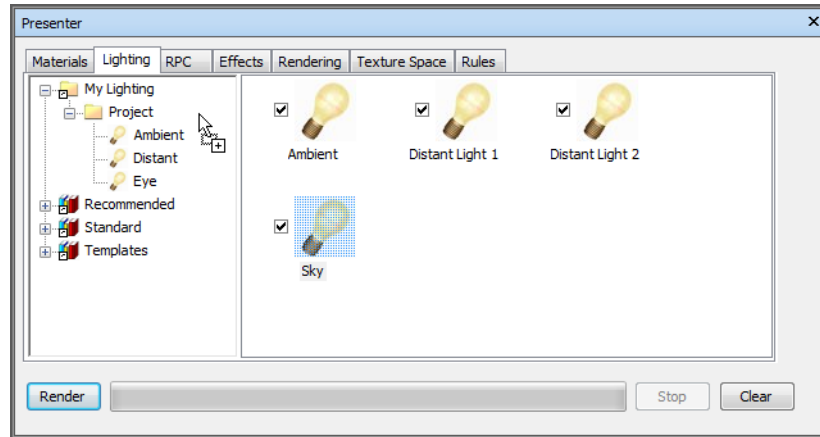
To delete a custom folder

- 1 In the **Presenter** window, **Lighting** tab, expand the **My Lighting** folder in the left pane, and right-click the folder you want to delete.
- 2 Click **Delete** on the shortcut menu.

To copy lighting effects into a custom folder

- 1 In the **Presenter** window, **Lighting** tab, click a lighting effect in the right pane, and drag it to the desired location under the **My Lighting** folder in the left pane until a mouse pointer displays a small plus symbol.

- 2 Release the left mouse button to drop the lighting effect into the folder.



NOTE You can also copy lighting effects between the folders in the left pane by either dragging and dropping them, or using the **Copy** and **Paste** options on the shortcut menu.

To manage palette lights

- 1 Right-click a light in the right pane of the **Lighting** tab (the palette).
 - Click **Copy** to copy the light to the clipboard. Right-click an empty space in the palette and click **Paste** to paste a copy of the light with the same name suffixed with the next number in the list.
 - Click **Delete** to delete the light from the palette. This will also remove the light from the scene.
 - Click **Rename** to rename the light. You can also select the light and press **F2** to rename it.
 - Click **Edit** or double-click a light to open the **Light Editor** dialog box, allowing you to edit its parameters. See [Edit Lights](#) (page 573) for more information.
 - Click **Clear Palette** to delete all the lights from the palette and hence from the scene.

Edit Lights

You can edit a light in the palette by double-clicking it, or right-clicking and choosing **Edit** on the shortcut menu.

There are six types of light visible in both OpenGL interactive renders and photorealistic renders:

- **Ambient** lights give a general background light to the scene and therefore only have **Intensity** and **Color** parameters.
- **Distant** lights are directional and so have a location and target. However, the location and target merely set up an axis down which the light shines, as these light types are infinitely far away and their beams are parallel. As well as **Intensity** and **Color** parameters, they can also cast shadows in a photorealistic render.
- **Eye** lights are located at the viewpoint and also only have **Intensity** and **Color** parameters.
- **Point** lights have a location but shine in all directions. They also have an **Intensity** and **Color** and additionally can cast **Shadows** (only available in a full photorealistic render).
- **Spot** lights are also directional and therefore have a location and target, as well as **Intensity** and **Color**, and shadow parameters. In addition, they also have parameters for affecting the light's **Fall Off** and **Cone Angle**, as these light types are not infinitely far away, so do spread their light over a cone and the intensity does diminish away from the light.
- **Sun** simulates the sun's light. The orientation of your model is defined by **North** and **Up** directions. The position of the sun is specified as **Azimuth** and **Altitude**. If the sun's **Mode** includes **Position**, you can input your location on earth, the **Time** (using local time zone) and date and **Presenter** will calculate the sun's azimuth and altitude for you. If the sun's **Mode** includes **Intensity**, **Presenter** will also calculate an accurate intensity for the sun based on position, time of year and atmospheric conditions.

There are an additional three types of light visible only in photorealistic renders:

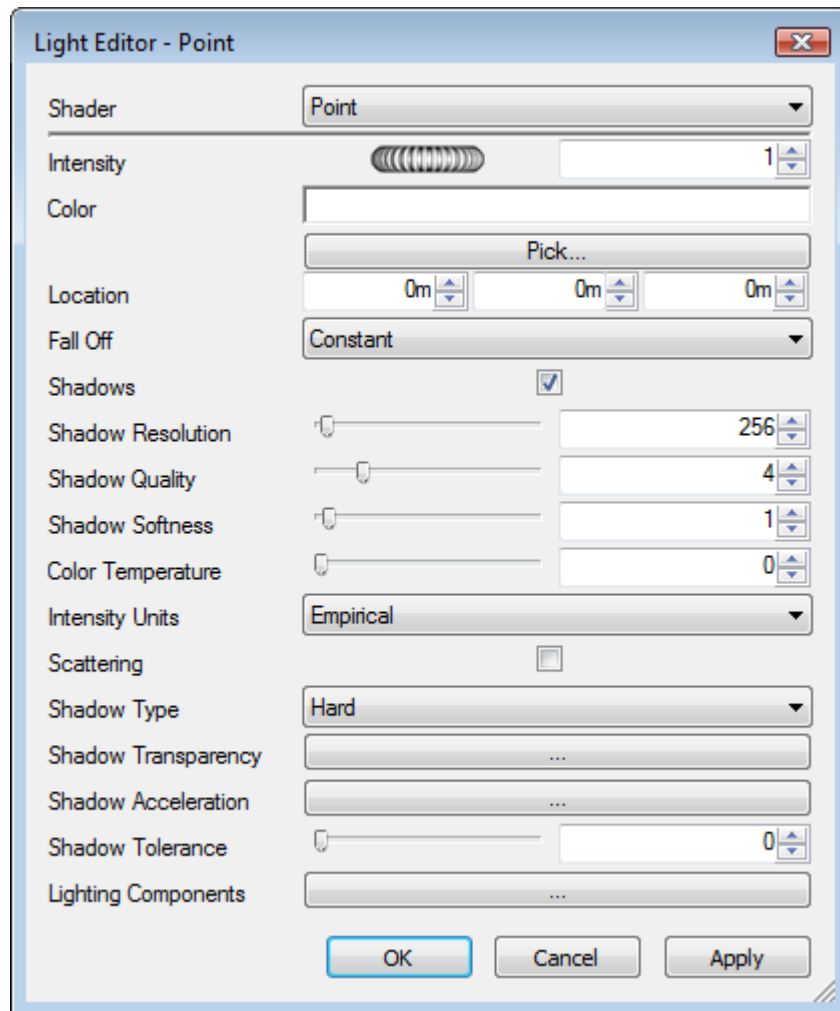
- **Projector** lights are used to project an image onto surfaces. You can define the file of the image to be projected.
- **Sky** simulates the illumination from the sky (but not the direct contribution due to the sun itself). The orientation of your model is defined by **North** and **Up** directions. The position of the sun is specified as **Sun Altitude** and **Sun Azimuth**. While the direct contribution of the sun is

not included, its location will determine the appearance of the sky hemisphere. If the **Intensity** is left at 0, **Presenter** will calculate an accurate intensity for you based on the sun's position.

- **Goniometric** lights can emit widely varying amounts of light energy in different directions. One goniometric source could behave exactly like a point light, another could behave exactly like a spot light, and a third could look nothing like either of those. A goniometric light gets its intensity distribution function (how much light goes in any one direction) from an industry-standard file. **Presenter** supports CIE, IES, CIB, and LDT files.

NOTE A complete reference manual for all light types is included with the Autodesk Navisworks API (see `\API\COM\documentation\shaders.chm`). The Autodesk Navisworks API is included with Autodesk Navisworks Manage 2012; it's an optional feature in the installer, and is installed by default into the API subfolder in the Autodesk Navisworks installation folder.

Each light type has its own parameters, and the editor for a **Point Light** is shown here:



Point, distant, spot and projector lights have a **Location** parameter. Distant and spot lights additionally have a **To** parameter. See [Add and Position Lights](#) (page 569) for more information.

NOTE If the **Presenter** profile in the **Options Editor** (see [Presenter Page](#) (page 824) for more information) is set to **Advanced**, the dialog box will include a full list of available parameters and allow you to change the type of a light.

Point, distant, spot, sky, sun, projector and goniometric lights have the **Shadows** parameter. See [Shadow Casting](#) (page 576) for more information.

Editing parameters in the dialog box will interactively alter the scene with those changes.

At any time, click **Apply** to apply the parameter edits to the light in the scene.

You can save an edited light for use in other scenes by dragging it onto the **My Lighting** user archive.

Shadow Casting

Selecting the **Shadows** check box in the **Light Editor** of a light that supports shadows (point, distant, spot, sky, sun, projector and goniometric) results in the selected light casting shadows in the scene. Shadows will only be visible in photorealistic renders, unless you are using a Hardware accelerated OpenGL 1.5 compliant graphics card, in which case, you can preview *interactive shadows*. See [Presenter Page](#) (page 824) for details on how to display interactive shadows and lighting.

NOTE Enabling shadows on lights should be given due consideration. If you turn on shadows on all lights, then you may find the effect very confusing and somewhat unnatural, especially if there are many lights in a small scene. This will also affect performance, during navigation and general refreshing of the **Scene View**. You may want to consider only enabling shadows on a few strategically positioned lights, to create the effect you require.

In addition to choosing which *lights* in your model will cast shadows, you may also select which *geometry items* in the model should cast a shadow. Each item in the scene has its own shadow casting option.

To set shadow casting from the Scene View

- Right-click an item in the **Scene View**, click **Presenter** on the shortcut menu, click **Shadows**, and click the shadow casting option you require. The available shadow casting options available for a geometry item are:
 - **On**. Choose this to enable shadows. The selected item will cast a shadow from any light source that has **Shadows** enabled.
 - **Off**. Choose this to disable shadows. The selected item will not cast a shadow from any light source.
 - **Inherit**. Choose this to inherit the shadow casting option from the parent item. That is, the selected item will use the same option as the item directly above it in the **Selection Tree** path (see [Selection Tree Window](#) (page 338) for more information). For example, if you turn

shadow casting **On** for a group, and the geometry contained within that layer is set to **Inherit**, then the geometry will also cast shadows, as it inherits the **On** option from its parent (the group).

NOTE If all items in the scene are set to **Inherit**, then the default setting is **On**.

NOTE The item selected will depend on your **Selection Resolution** setting. See [Set Selection Resolution](#) (page 345) for more information.

To set shadow casting from the Selection Tree

- Right-click an item in the **Selection Tree**, click **Presenter** on the shortcut menu, click **Shadows**, and click the shadow casting option you require.

The available shadow casting options available for a geometry item are:

- **Off**. Choose this to disable shadows. The selected item will not cast a shadow from any light source.
- **On**. Choose this to enable shadows. The selected item will cast a shadow from any light source that has **Shadows** enabled.
- **Inherit**. Choose this to inherit the shadow casting option from the parent item. That is, the selected item will use the same option as the item directly above it in the **Selection Tree** path (see [Selection Tree Window](#) (page 338) for more information). For example, if you turn shadow casting **On** for a group, and the geometry contained within that layer is set to **Inherit**, then the geometry will also cast shadows, as it inherits the **On** option from its parent (the group).

NOTE If all items in the scene are set to **Inherit**, then the default setting is **On**.

Advanced Lighting

You can use **Presenter** to apply advanced lighting effects.

Soft Shadows

Presenter includes shadows generated from pre-calculated, shadow maps for each shadow-casting light source. The use of shadow maps enables rapid rendering of shadows with soft or graduated edges. The shadow resolution can be controlled to balance performance and image quality.

Soft shadows are only suitable for use with small models, and are disabled by default. For large models, the generation of the shadow maps can use excessive amounts of time and memory. Soft shadows generated for large models are often too vague and dispersed without using an excessively high resolution, which uses even more memory and time.

Physically Accurate Lights

By default, **Presenter** uses lights with unitless, or empirical intensities. These are physically meaningless, and are just chosen to give a visually pleasing result. **Presenter** can also use physically accurate intensities. These are defined in real-world units such as Candela, Lumen, or Lux. However, once you start using lights with real-world intensities, you start to produce images with a real-world variation in luminance values.

By default, **Presenter** uses lights whose intensity remains constant as you move further from the light. In the real-world intensity is reduced proportional to the inverse square of the distance from the light. Changing a light's **Fall Off** parameter to **Inverse Square Law** will more accurately model a light's fall off in intensity. However, once you start using lights with real-world fall off, you start to produce images with a real-world variation in luminance values.

In the real world, the human eye is capable of resolving images in extremely varied lighting conditions, ranging from bright sunshine reflecting off snow to a room lit only by a single candle. In computer graphics, however, you need to produce an image on a display device which has a very limited range of luminance values. Therefore, it is necessary to compress the range of luminance values found in a real-world scene into the displayable range in such a way as to produce a realistic looking image.

Photography, of course, has exactly the same problem. If a photographer (or camera) does not take into account the light levels in a scene before calculating the exposure of the shot, the likely result will be an image which is either over-exposed (everything is too bright) or under-exposed (everything is too dark). A professional photographer will also use different speeds of film for

different lighting conditions. The aim is to produce an image on film that is representative of how that scene would have looked to a human observer.

Presenter includes the **Auto Exposure** option (see [Auto Exposure](#) (page 598)). When enabled, **Presenter** will render the image twice. Once to sample the range of luminance values in the output image, then a second time to render the actual image with the luminance values adjusted to match the behavior of the human eye.

In general, when using physically accurate lights, the **Auto Exposure** option should be on.

Volumetric Lights

Volumetric lighting allows effects such as the scattering of light, by fog or smoke, in a scene. To use this effect, select the **Scattering** check box on each light. A **Scattering Medium** foreground effect must also be in use (see [Foreground Effects](#) (page 593) for more information).

NOTE You may need to adjust the **Medium Density** and **Medium Ambient** parameters of the **Scattering Medium** foreground effect to suit your model. If no volumetric effects are visible, the **Medium Density** is too low. If the rendered image is entirely white, the **Medium Density** is too high.

The default medium is plain white. Optionally, a density shader may be set to any solid (not wrapped) color shader, to create the effect of a non-uniform (inhomogeneous) medium. Examples of shaders that can be used are **Blue Marble** and **Solid Clouds**. A shader that has been designed explicitly for this purpose is the **Turbulent** shader.

The key points when using volumetric lighting are:

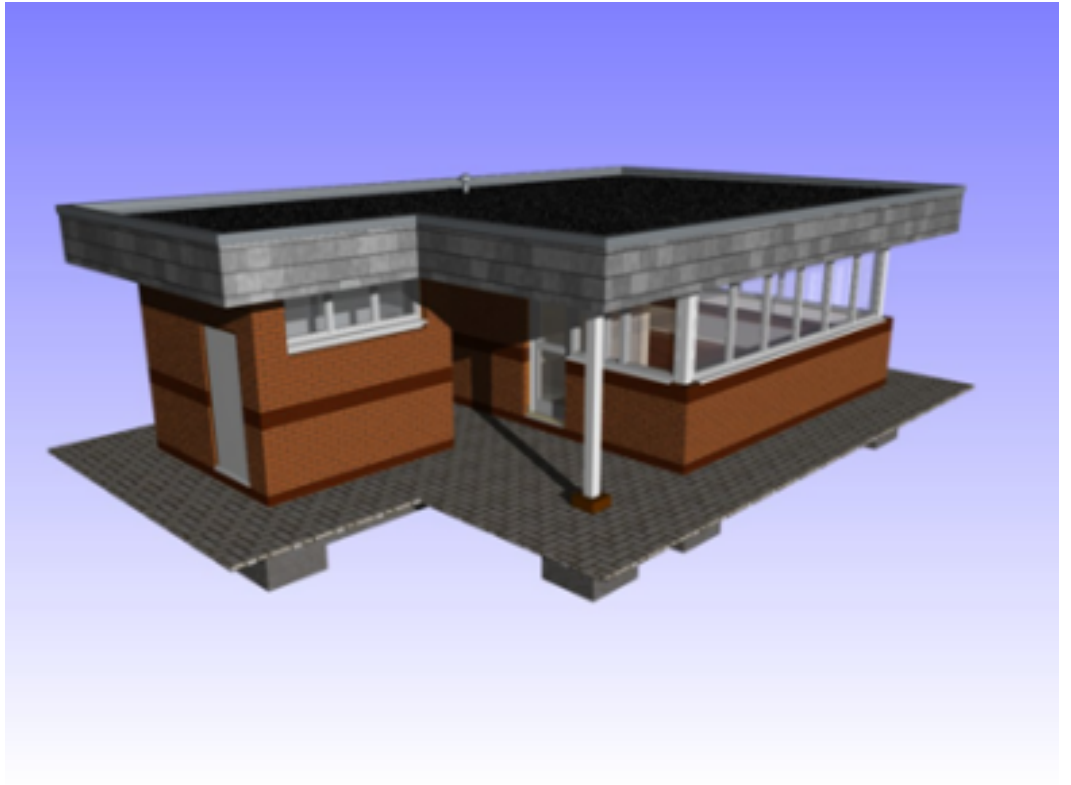
- Remember to turn the **Scattering** parameter of light sources on if you want to see their volumetric effects.
- Use **Medium Density** and **Medium Color** to define brightness and color of the lit medium.
- Use a solid color shader set as density shader for simulation of density variations in the medium.
- Decrease the **Error Bound** parameter, if image appears spotty outside shadow areas.
- Increase the **Min LOD** parameter, if areas with volumetric shadows appear spotty.

- For fast previews, set the **Error Bound** parameter high, and the **Min LOD** parameter small.
- For best results, set the **Fall Off** option to **Inverse Square Law**, and turn on **Auto Exposure**.

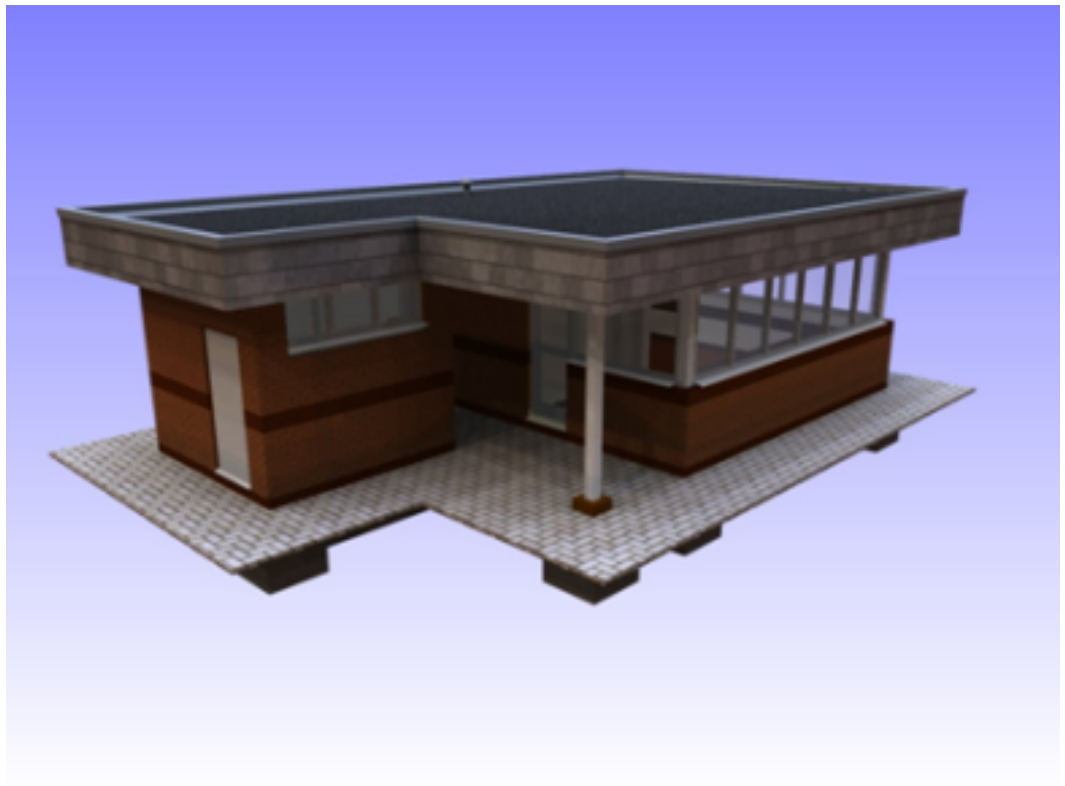
Image-based Lighting

Image-based lighting is where an image is used to light a scene. In the real world, every object is lit not only by light sources like the sun, lamps, and so on, but also by everything around. Standing in the middle of a street, a person will be lit by the sun, the blue sky, the brown buildings and the grey street. Emulating this form of lighting clearly has the potential to create incredibly realistic images.

Images used in this lighting method are a special kind of image, called a High Dynamic Range Image or HDRI. This type of image has the capability of lighting a scene with incredible accuracy. In the **Presenter** window, an HDRI is wrapped around the scene as a sphere, and color and brightness from the HDRI are cast onto the 3D model to light it.



A model lit with normal lights



The same model lit with image-based lighting

You can see the difference this form of lighting can make to rendered images. And the enormous advantage here is that it is much easier to set up than traditional lighting.

To set up image-based lighting

- 1 Open the **Presenter** window, and click the **Lighting** tab.
- 2 Expand the **Recommended** archive in the left pane of the **Lighting** tab.
- 3 Drag the **Environment Light Studio** into the palette on the right. This replaces all lights that were in the palette with an **Ambient** and an **Environment** light containing a **High Dynamic Range Image**.
- 4 Click **Render** to render the scene using the default image contained in this **Environment** light. This type of render can take slightly longer

than traditional lighting methods, but the results are worth the extra time invested.

- 5 To use an alternative sample image, expand the **Environment** folder in the left pane to view another two example environment lights; **Sky** and **City**. Drag the **City** light into the palette on the right to replace the **Environment** light in the palette.

NOTE Before rendering, either delete the **Environment** light from the palette, or deselect the check box.

To manually insert a new HDRI

- 1 Open the **Presenter** window, and click the **Lighting** tab.
- 2 Right-click the **Environment** light in the palette on the right, and click **Edit** on the shortcut menu.
- 3 In the **Light Editor**, click **Edit** in the **Environment** field.
- 4 In the **Shader Editor**, click the **Browse** button (...) in the **File Name** field, and browse to the required HDR file.

NOTE For this to work correctly, this HDRI must be a **Light Probe** HDRI. Additional Light Probe HDRIs are available from a variety of places on the Internet, including [Dosch Design](#).

Click on **OK** in both dialog boxes to set the new image to be the light source.

- 5 In the **Presenter** window, click **Render** to produce a newly lit rendered scene.

Use Presenter RPCs

RPC Tab

The RPC (Rich Photorealistic Content) support in the **Presenter** window enables the addition of photographic scenery to any 3D project. RPC files can be bought directly from [ArchVision](#), and typically come in libraries of content ranging from trees and plants to people. They also come in a variety of types.

- **2D** RPCs are single-direction 2D photographs that always face the camera, and are a single frame, looking the same from every angle, and not animating.

- **3D** RPCs are objects that have a high number of frames allowing the camera to move around the object and see it from all angles.
- **2.5D** RPCs are animated, single-direction 2D photographs. Animated RPCs will only animate visually when exported as a rendered animation.
- **3.5D** RPCs include animation and views from all around the object.
- **3D+** RPCs, often called smart content, are not currently supported.



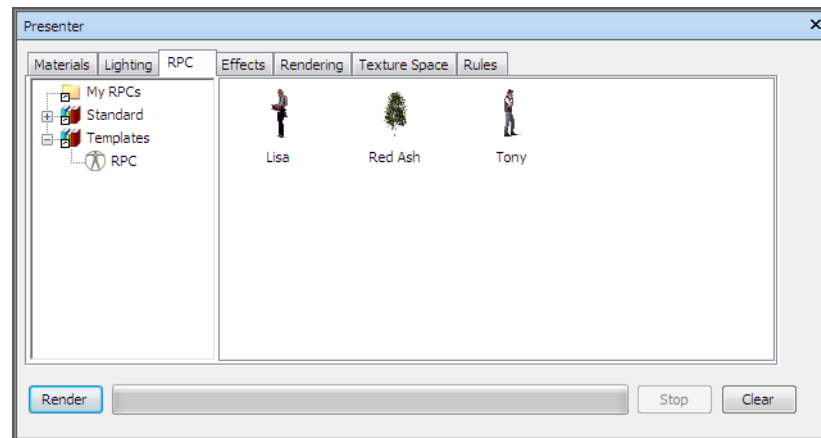
An example of RPC people on and around a building

The key benefit to using RPC content is that it only takes a short amount of time to fill a scene with realistic content, and it adds very little to the rendering time.

NOTE While navigating through the scene, RPCs will always turn to face the camera. If an RPC is 3D or 3.5D, right-clicking it in the **Scene View**, and clicking **Refresh** on the shortcut menu will set it to the correct frame, based on the current camera position. Rendering the scene will always refresh all RPCs.

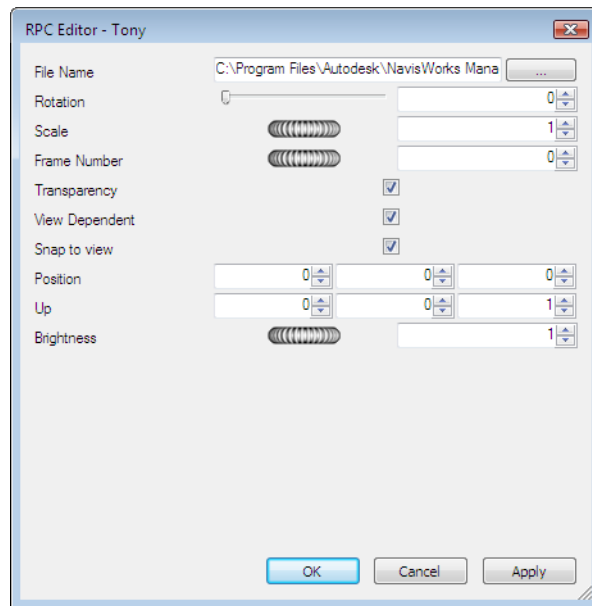
IMPORTANT When publishing a scene using the **Publisher** dialog box, any RPCs included in that scene are not published to the **_Presenter_Maps** directory nor embedded in the NWD file. The size of the files, and the fact that most RPCs are licensed, currently prohibits this.

Like the [Materials](#) (page 556), [Lighting](#) (page 568), and [Rendering](#) (page 594) tabs, the **RPC** tab is divided into two panes, with the archives on the left and the palette on the right. This tab is where you set up **RPCs**.



To add an RPC to a palette

- 1 Open the **Presenter** window, and click the **RPC** tab.
- 2 In the left pane, expand the **Templates** archive, and drag the RPC icon into the palette on the right.
- 3 Double-click the RPC icon in the palette.
- 4 In the **RPC Editor**, click the **Browse** button (...) in the **File Name** field.



- 5 In the **Open RPC File** dialog box, locate the desired RPC file, and click **Open**. Autodesk Navisworks has a small selection of free, example RPC files in the resources area of the product DVD. You can use them to make up a simple scene with some people, plants, and trees.
- 6 Adjust the settings in the **RPC Editor** as needed, and click **OK**.

To add an RPC to a model

- 1 Open the **Presenter** window, and click the **RPC** tab.
- 2 To add an RPC to the model, either right-click desired RPC icon in the palette and select **Add Instance**, which gives a target mouse pointer with which to and click on a location in the **Scene View**, or left-click and drag the RPC icon from the palette onto the desired position in the **Scene View**.
- 3 Click **Render** to see how the RPC effect looks in the scene.

To move an RPC

- Right-click the RPC you want to move in the **Scene View**, and click **Pick Position** on the shortcut menu. This changes the mouse pointer to a target for selection of an alternative location.

To edit an RPC

- 1 Open the **Presenter** window, and click the **RPC** tab.
- 2 Double-click the required RPC in the palette.
- 3 Use the **RPC Editor** to adjust the settings as needed. For example, you can use **Rotation** to set which direction people are facing, or you can change the **Scale** size.
- 4 Click **OK**.

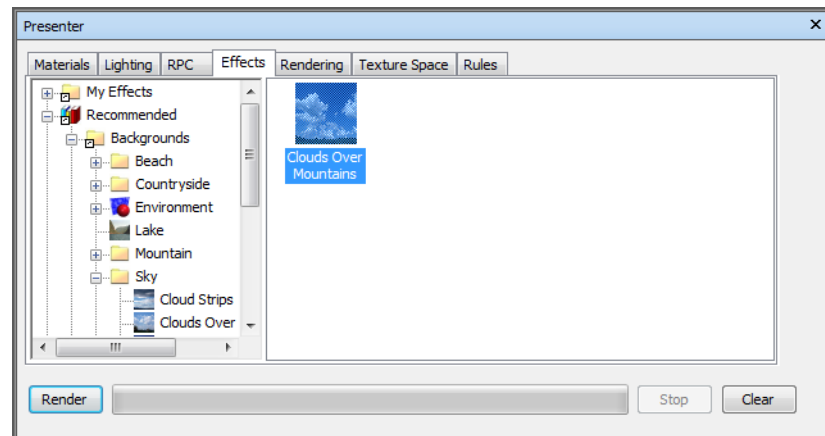
To delete an RPC

- Right-click the RPC you want to delete in the **Scene View**, and click **Delete** on the shortcut menu.

Use Presenter Rendering Effects

Effects Tab

Like the [Materials](#) (page 556), [Lighting](#) (page 568), and [Rendering](#) (page 594) tabs, the **Effects** tab is divided into two panes, with the archives on the left and the palette on the right. This tab is where you set up different background and foreground effects.



You can only have one background and one foreground effect at a time in the palette.

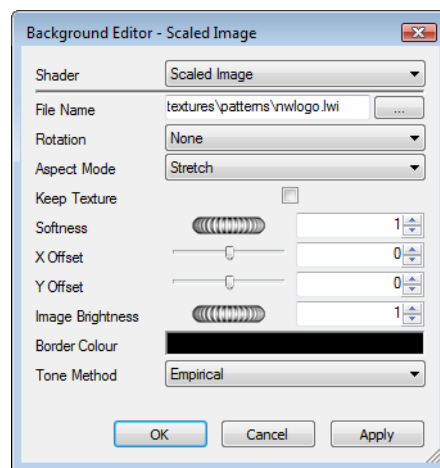
NOTE A complete reference manual for all types of foregrounds, backgrounds and rendering styles is included with the Autodesk Navisworks API (see `\API\COM\documentation\shaders.chm`). The Autodesk Navisworks API is included with Autodesk Navisworks Manage 2012; it's an optional feature in the installer, and is installed by default into the API subfolder in the Autodesk Navisworks installation folder.

Background Effects

Background effects change the background of the image when rendered and include plain colors, graduated colors, procedural clouds, and image files (tiled or scaled).

Most backgrounds can be rendered interactively in OpenGL so you have a good preview of how the background will be fully rendered.

Double-click an effect in the palette to open the **Background Editor**. Each editor will be different for each type of background. The **Scaled Image** editor is shown here:



NOTE If the **Presenter** profile in the **Options Editor** (see [Presenter Page](#) (page 824) for more information) is set to **Advanced**, the dialog box will include the full list of parameters and allow you to change the type of background.

In the **Background Editor**, you can change backgrounds by clicking the **Browse** button (...) next to the **File Name** field and open a new image as the background.

Editing parameters in the **Background Editor** will interactively alter the scene with those changes. At any time, click **Apply** to apply the parameter edits to the scene.

You can save an edited background for use in other scenes by dragging it into the **Backgrounds** folder inside the **My Effects** user archive.

Environment Background and Reflections

Environment backgrounds are a special kind of background that move with the model and allow reflections from reflective model parts.

Environment backgrounds not only make the scene look more realistic for static images of the model, but completely transform the realism effect in animations. This is because as in real life, by moving an object around in the background, the environment also changes. For example, looking up to the top of a building, the background will be the sky, whereas, an aerial view of the building will have the surrounding buildings or landscape as the background.



Another benefit of using environment backgrounds is that reflection properties of materials in the model are able to reflect the environment (for example, when looking up, towards the top of a glass-fronted building, the sky is reflected in the glass).



Environment backgrounds are images wrapped around the model. The recommended images to use are vertical cross maps. These images are automatically placed on the inside of a cube that then surrounds the scene. Due to the nature of the feature, edges and corners become indistinguishable. A wide variety of environment maps are available for purchase on the Internet.

Setting up an environment background consists of two components: the background image, such as the sky over the desert, and the environment that holds the background image.

To add a background effect

- 1 Open the **Presenter** window, and click the **Effects** tab.
- 2 In the left pane, expand the **Recommended** archive, and choose the desired background effect.

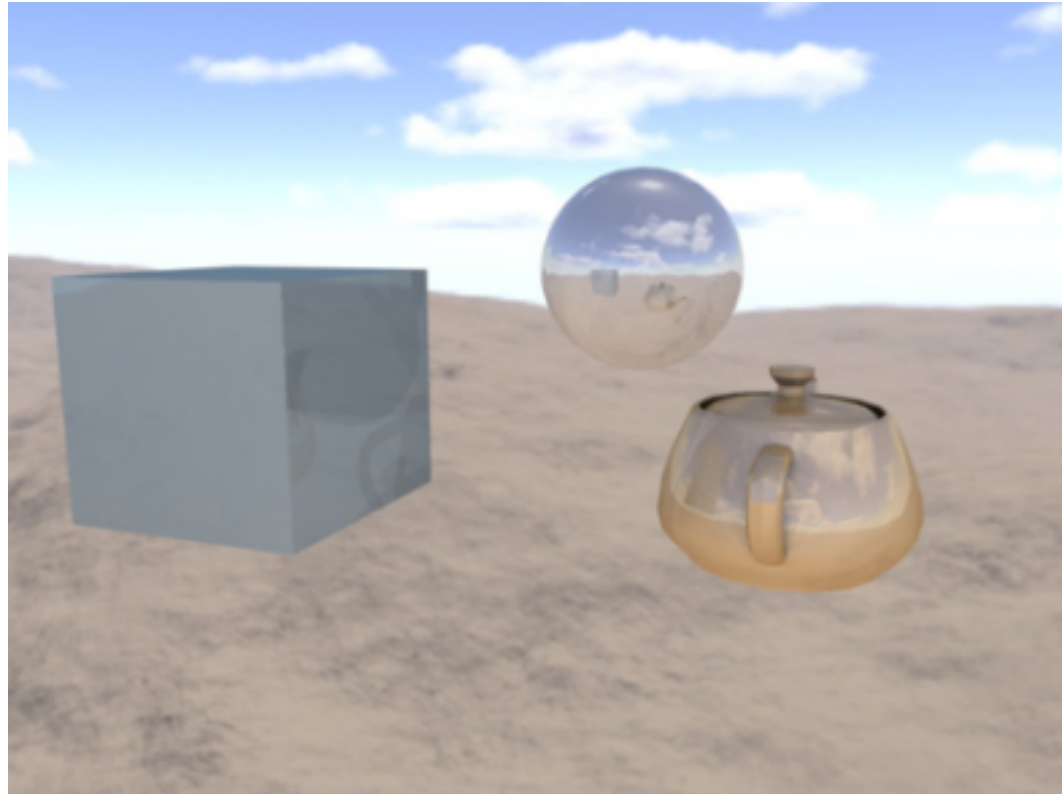
- 3 Drag your chosen background into the palette on the right side.
Most backgrounds can be rendered interactively in OpenGL so you have a good preview of how the background will be fully rendered.
- 4 Click **Render** to apply the background to the model.

To edit a background effect

- 1 Open the **Presenter** window, and click the **Effects** tab.
- 2 Double-click the required background in the palette.
- 3 Use the **Background Editor** to adjust the settings as needed.
Each editor will be different for each type of background.
- 4 Click **OK**.

To add an environment background

- 1 Open the **Presenter** window, and click the **Effects** tab.
- 2 In the left pane, expand the **Recommended** archive, open the **Environments** subfolder, and then open the **Panorama** folder.
- 3 Drag the **Sky** effect over to the palette on the right.
The background is not yet visible in the **Scene View**.
- 4 Return to the **Recommended** archive, open the **Backgrounds** subfolder, and drag the **Environment** effect over to the palette.
The background is now visible. This is because this type of background is made up of two parts, the **Sky** image and the background **Environment** shader that points at the image; the two elements link together automatically.



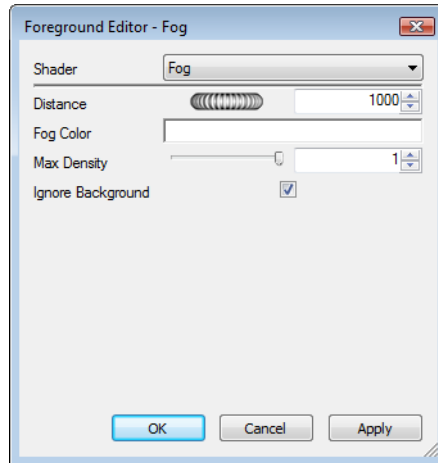
Once this is complete, moving around the model will see the background moving appropriately in real-time. With a modern graphics card, if there are reflective surfaces in the scene, setting the **PresenterHardware Shading** in the **Options Editor** to **Lighting** will show the newly set up background reflected in all such surfaces (see [Presenter Page](#) (page 824)).

TIP Click **Render** to show this background in the photorealistic output.

Foreground Effects

Foreground effects change the foreground of the image when rendered, and include fog and snow effects. None of these effects are available as an interactive preview and can only be seen when a full render is done.

Adding foreground effects is similar to adding background effects. Double-click an effect in the palette to open the **Foreground Editor**. Each editor will be different for each foreground type. The **Fog** editor is shown here:



NOTE If the **Presenter** profile in the **Options Editor** (see [Presenter Page](#) (page 824) for more information) is set to **Advanced**, the dialog box will include the full list of parameters and allow you to change the type of foreground.

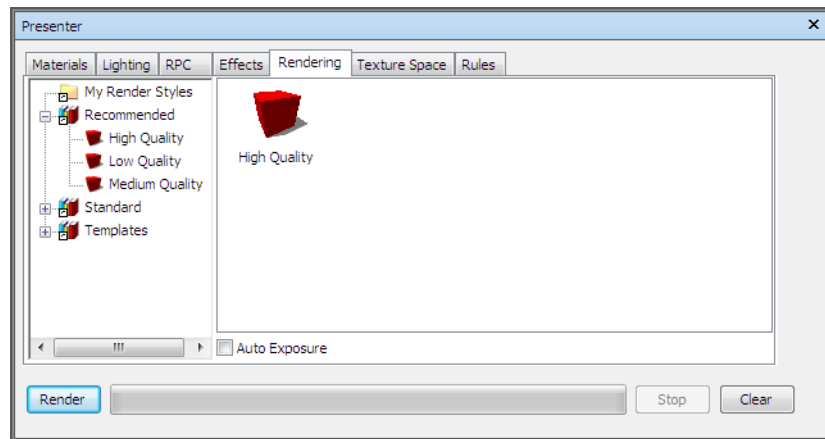
At any time, click **Apply** to apply the parameter edits to the scene.

You can save an edited foreground for use in other scenes by dragging it into the **Foregrounds** folder inside the **My Effects** user archive.

Use Presenter Rendering Styles

Rendering Tab

Like the [Materials](#) (page 556), [Lighting](#) (page 568), and [Effects](#) (page 587) tabs, the **Rendering** tab is divided into two panes, with the archives on the left and the palette on the right. This tab is where you select in which style and how you want the scene to be rendered. Each archive has a number of different render styles to choose from.



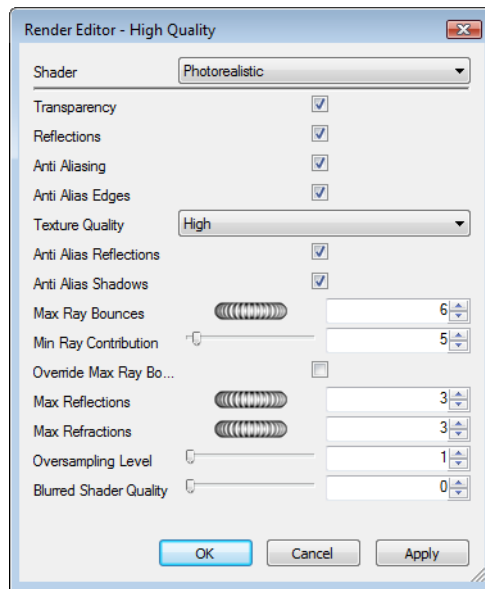
NOTE A complete reference manual for all types of rendering styles is included with the Autodesk Navisworks API, (see \API\COM\documentation\shaders.chm). The Autodesk Navisworks API is included with Autodesk Navisworks Manage 2012; it's an optional feature in the installer, and is installed by default into the API subfolder in the Autodesk Navisworks installation folder.

Rendering Styles

Rendering styles affect the way that the scene is rendered when a full photorealistic render is done (by clicking **Render**). None of these effects are available as an interactive preview.

To set up a rendering style, drag your chosen style into the palette on the right side of the tab.

To edit your chosen rendering style, double-click the style in the palette. The **Render Editor** dialog box opens. Each editor will be different for each type of rendering style. The **High Quality** editor is shown here:



NOTE If the **Presenter** profile in the **Options Editor** (see [Presenter Page](#) (page 824) for more information) is set to **Advanced**, the dialog box will include the full list of parameters and allow you to change the type of foreground.

At any time, click **Apply** to apply the parameter edits to the scene.

You can save an edited rendering style for use in other scenes by dragging it onto the **My Render Styles** user archive.

Predefined Rendering Styles

The **Recommended** archive contains three predefined rendering styles:

- **High Quality.** Choose this rendering style for the highest quality rendered output. This includes all reflections and transparencies and anti-aliasing on edges, reflections and shadows. Of the three recommended rendering styles, this will take the longest to render. This is also the rendering style used if no other is chosen. Use this style for the final export of your rendered output.
- **Low Quality.** Choose this rendering style for a quick, low-quality render. This includes no reflections or anti-aliasing. Use this style if you want to quickly see the affect of materials and lighting you have applied to the scene.

- **Medium Quality.** Choose this rendering style for a medium-quality render. This includes all reflections and transparencies and anti-aliasing only on shadows. You may use this style for a final preview of the scene, prior to exporting your final rendered output.

The **Standard** archive contains many rendering styles that simulate hand drawing and other non-photorealistic styles. These styles use a mixture of shaded, vector, and image based rendering techniques. They are generally best suited to small models and small output images.

NOTE The **Standard** rendering styles require multiple stages to render a scene. Therefore, these styles can often take a considerable time to render.

The **Templates** archive contains five main types of rendering style, which can be used to define your own rendering styles:

- **Photorealistic (Raytrace).** This archive contains photorealistic rendering styles, including **High Quality**, **Low Quality** and **Medium Quality** like the **Recommended** archive. These rendering styles are the fastest and use the least memory where large parts of the model are obscured from any particular viewpoint. For example, when inside a room within a building, the walls of the room will obscure the rest of the building from view.
- **Photorealistic (Scanline).** This archive contains photorealistic rendering styles, including **High Quality**, **Low Quality** and **Medium Quality** like the **Recommended** archive. These rendering styles are the fastest and use the least memory where most of the model is visible from any particular viewpoint. For example, when rendering an overview of a plant and process model, the majority of the model can be seen as there are fewer walls, or similar, to obscure your view.
- **Simple Shaded.** This template is a simple, shaded rendering style, where advanced features, such as textures and transparency, are not required.
- **Sketch.** This archive contains many basic sketch rendering styles.

NOTE **Sketch** rendering styles require multiple stages to render a scene. Therefore, these styles can often take a considerable time to render.

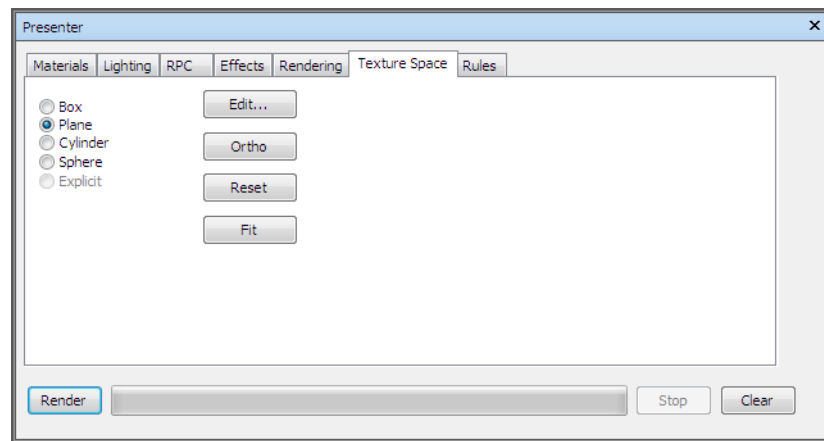
- **Vector.** This template is a vector rendering style, which will render the scene in wireframe.

Auto Exposure

Select the **Auto Exposure** check box to render the scene with balanced brightness and contrast. This is essential when using physically accurate lighting, such as a **Sky** or **Sun** light. If adding either of these lights to your scene, you will be prompted to turn **Auto Exposure** on, if it is not already.

Use Presenter Texture Space

Texture space describes the way in which a texture is applied to an item. For example, applying a cylindrical texture space to a pipe will cause textures on the pipe to be rendered more naturally. An item's texture space may have been assigned from the original CAD application and stored in the native CAD file, or set up within the **Presenter** window with the options of **Box**, **Plane**, **Cylinder**, or **Sphere**. The **Explicit** option allows a user-defined texture space to be applied and will be available if the item had a texture space applied to it in the original CAD application. Each texture space option applies some imaginary bounding geometry around the item and “shrink-wraps” the texture as best it can to the geometry underneath this bounding geometry.



If you've applied a texture to an item, the **Presenter** window will attempt to work out the best fit from the four texture spaces available. If this isn't what you intended, then you can then edit the texture space.

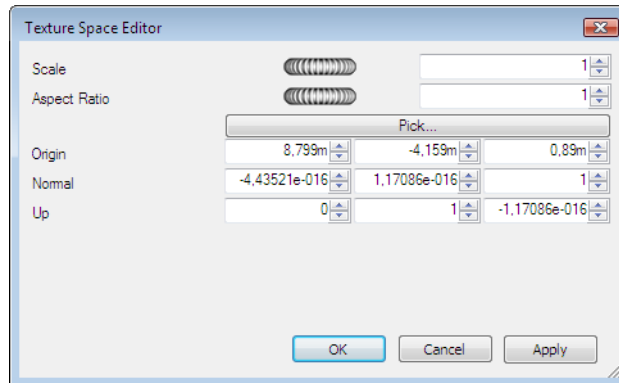
To edit texture space

- 1 Select the item (see [Interactive Geometry Selection](#) (page 337) for more information) and go to the **Texture Space** tab in the **Presenter** window to choose another texture space.

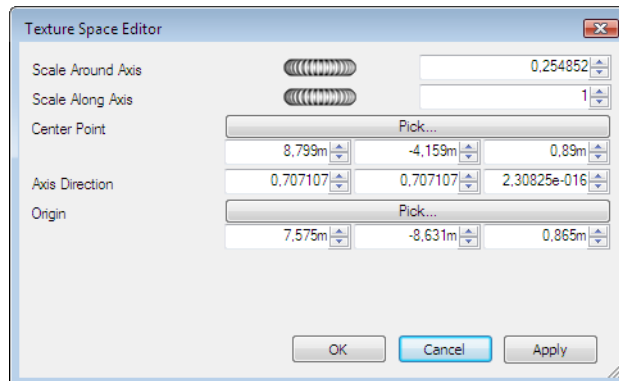
Alternatively, right-click an item which has a texture applied to it and click a new texture space on the **Presenter** ► **Texture Space** shortcut menu.

- 2 You can fine-tune an item's texture space further by clicking **Edit** on the **Texture Space** tab of the **Presenter** window. The relevant **Texture Space Editor** opens.

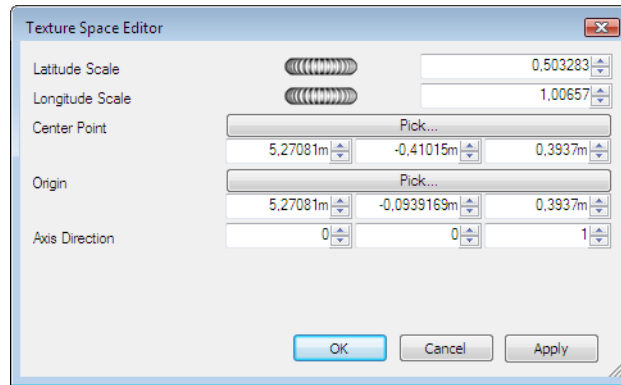
Either the plane texture space editor:



Or the cylindrical texture space editor:



Or the spherical texture space editor:

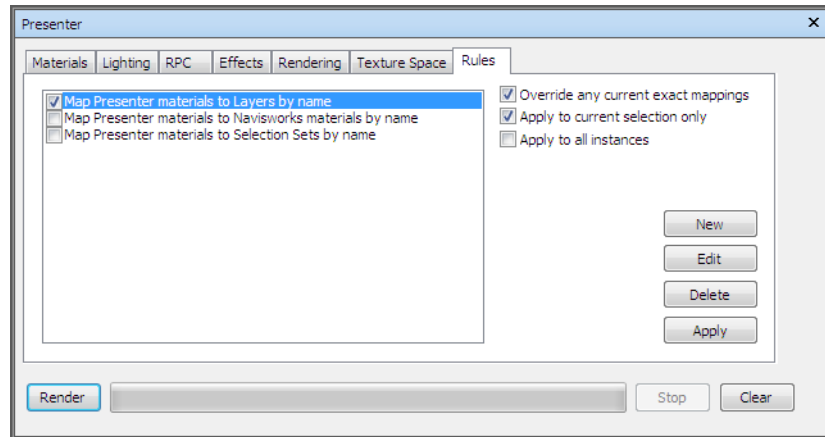


- 3 Edit each of the individual parameters and click **Apply** to see the results of the edit. Clicking **Pick** allows you to interactively pick a point in the model rather than individually typing in X-, Y-, and Z- coordinates.
- 4 Click **OK**.
- 5 Once edited, you can always reset a texture space to the Autodesk Navisworks-defined version by clicking **Reset** on the **Texture Space** tab.
- 6 Click **Ortho** if you want Autodesk Navisworks to align the texture space with the nearest axis.
- 7 Click **Fit** if you want Autodesk Navisworks to fit a single repeat of the texture to the item. This is of particular use when you are mapping an image to the exact size of the item it's being applied to.

Use Presenter Rules

Rules Tab

The **Rules** tab of the **Presenter** window allows you to apply materials to models according to certain user-defined criteria, rather than by dragging and dropping onto individual layers, groups or components. For example, all layers that are floors can be assigned a material called “floorboards”, without having to manually drag and drop the material on to each floorboard. Each time the model is updated, the rules then just need re-applying, rather than manually re-applying materials to all items.



All materials can be saved in a Autodesk Navisworks “palette” NWP file, which allows you to set up a palette of materials once for a project and re-apply them to a model as it evolves, or to another model in the same project that has been set up with the same layer names, colors, selection sets and/or properties.

See [Organize and Manage Materials](#) (page 559) for more information.

Predefined Rules

You can apply as many rules at once, as all rules depend on material names. Rules can be defined using the Autodesk Navisworks API, but the predefined rules are:

- **Layers by Name** enables you to apply a material to named layers. For example, if a layer is called “Doors” and you rename a material to “Doors” (the spelling and case must be exactly the same as the layer’s name) then all layers named “Doors” will display the properties of that material when you select the rule **Layers by Name**, and click **Apply**. You can give multiple materials the name of different layers and apply this rule to all of the layers.
- Autodesk Navisworks **Materials by Name** enables you to apply a material to named Autodesk Navisworks materials. Autodesk Navisworks materials are *not* the same as Autodesk Navisworks **Presenter** materials. Autodesk Navisworks materials merely refer to the color and transparency of the item as saved in the original CAD file, whereas **Presenter** materials are those materials applied using the **Presenter** tool, or are more complex materials, such as bitmaps converted from the original CAD file.

If a **Presenter** material has the same name as a Autodesk Navisworks material in the model (for example, “AutoCAD Color Index 7”), then all items with this original Autodesk Navisworks material name in the scene receive this **Presenter** material from the palette when you select the rule Autodesk Navisworks **Materials by Name**, and click **Apply**.

- **Selection Sets by Name** enables you to apply a material to selection sets. See [Create and Use Sets of Objects](#) (page 360) for more information on setting up selection sets.

If a material has the same name as a selection set in the model, then all items in this selection set receive this material from the palette when you select the rule **Selection Sets by Name**, and click **Apply**.

To apply predefined Presenter rules

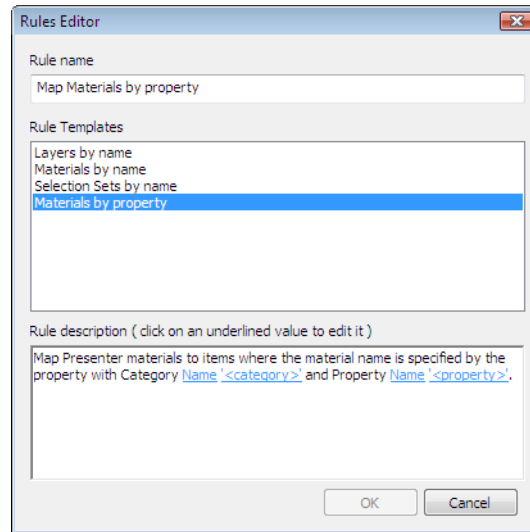
- 1 Open the **Presenter** window, and click the **Rules** tab.
- 2 Select all the rules you want to apply. You can apply as many rules as you want, as all rules depend on material names.
- 3 Use the check boxes on the right side of the **Presenter** window, **Rules** tab, to control how the selected rules applied to the scene:
 - Select **Override Any Current Exact Mappings** if you want to override any existing mappings from **Presenter** materials to items in the selection tree.
 - Select **Apply to Current Selection Only** if you want to apply this rule only to the currently selected items in the scene. Be aware that this is the default setting, so if your rule seems not to have worked, check that you don’t have this option checked with nothing selected in the scene.
 - Select **Apply to All Instances** if you want to apply this rule to all instances of any multiply-instanced item affected by the rule.
- 4 Click **Apply**.

Custom Rules

New custom rules may also be added and defined, using the **Rules Editor**.

To add custom Presenter rules

- 1 From the **Rules** tab of the **Presenter** window, click **New**.
The **Rules Editor** dialog box opens.



- 2 Enter a new name for your rule in the **Rule name** box.

NOTE If you choose not to enter a name, upon selecting a rule template, the name of that template will be used.

- 3 From the **Rule Templates** list, choose a template from which your rule will be based upon.

NOTE The **Layers by Name**, **Materials by Name**, and **Selection Sets by Name** templates are those used for the predefined **Presenter** rules.

The **Materials by Property** template enables you to specify a property within the model scene. If a material has the same name as the specified property value in the model, then all items having that property will receive this material from the palette when you select the rule **Materials by Property**, and click **Apply**.

- 4 In the **Rule description** box, click on each of the *underlined values* to define your custom rule. The customizable values available with the built in templates are:
 - **Name**. Use the name of the category or property as it is displayed in the interface (recommended). You can also choose **Internal Name**, which is accessed via the API (for advanced use only).
 - **'<category>'**. Choose from the available list, which category the property you want to define is in. Only the categories that are contained in the scene are available in the drop down.

- '<property>'. Choose from the available list, which property you want to define. Again, only the properties in the scene within the chosen category will be available.

5 Click **OK**.

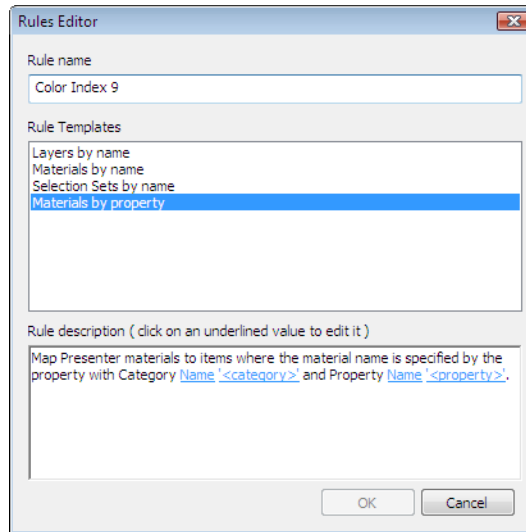
The Presenter Rules Example

To follow this procedure, open the file called `gatehouse_pub.nwd` located under the `Examples\Gatehouse` folder in the Autodesk Navisworks installation directory.

Let's assume, you want to apply window glass material to all gatehouse items with material property of **AutoCAD Color Index 9**, and red brick material to all gatehouse items with material property of **AutoCAD Color Index 32**. The following procedure describes the necessary steps to achieve this, using the **Presenter** rules.

To apply materials with the custom Presenter rules

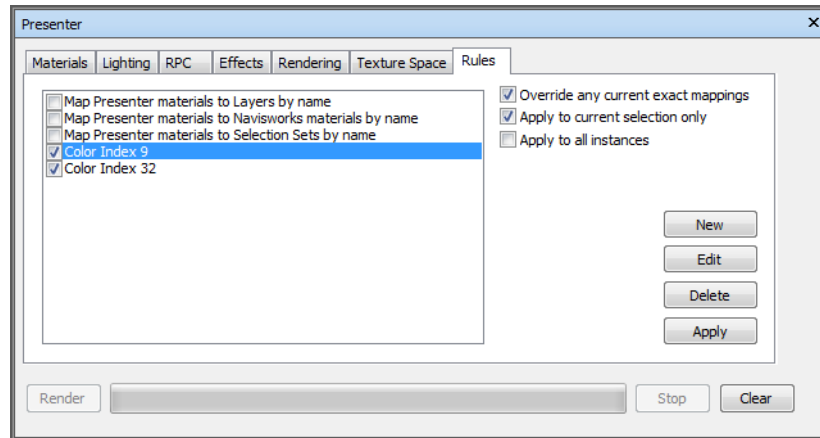
- 1 Open the **Presenter** window, and click the **Rules** tab.
- 2 Create a rule called "Color Index 9", based on the materials by property template:
 - a From the **Rules** tab of the **Presenter** window, click **New**.
 - b Enter the rule name "Color Index 9".
 - c Choose the **Materials by property** template.



- d Click '<category>' and choose **Material** from the drop-down list and click **OK**.
 - e Click '<property>' and choose **Name** from the drop-down list and click **OK**.
 - f Click **OK** to save the custom rule.
- 3 Create another rule called “Color Index 32” using the same settings as before.
 - 4 Set up the necessary materials:
 - 1 From the **Recommended\Glass** archive on the **Materials** tab, drag the **Window Glass** material into the palette, press **F2**, and rename it as “AutoCAD Color Index 9”. Press **Enter** to save the new name.

NOTE The name of the **Presenter** material must match the name of the material saved in the original CAD file. You can view the material properties in the **Properties** window or on the **Properties** tab of the **Selection Tree**.
 - 2 From the **Recommended\Bricks** archive on the **Materials** tab, drag the **Red Brick** material into the palette, press **F2**, and rename it as “AutoCAD Color Index 32”. Press **Enter** to save the new name.
 - 5 Click the **Rules** tab and select the **Color Index 9** and **Color Index 32** rules.

- 6 Ensure that only **Override Any Current Exact Mappings** is selected and click **Apply**, to apply the rules.



All items within the model scene that have the AutoCAD Color Index 9 and AutoCAD Color Index 32 properties should now have the corresponding **Presenter** materials applied to them.

Simulate Construction Scheduling

14

The **TimeLiner** tool enables you to link your 3D model to an external construction schedule for visual 4D planning.

Overview of TimeLiner Tool

The **TimeLiner** tool adds 4D schedule simulation to Autodesk Navisworks Manage 2012. **TimeLiner** imports schedules from a variety of sources. You can then connect tasks in the schedule with objects in the model to create a 4D simulation. This allows you to see the effects of the schedule on the model, and compare planned dates against actual dates. **TimeLiner** also allows the export of images and animations based on the results of the simulation. **TimeLiner** will automatically update the simulation if the model or schedule changes.

You can combine the functionality of **TimeLiner** with other Autodesk Navisworks tools:

- Linking **TimeLiner** and Object Animation together enables the triggering and scheduling of object movement based on start time and duration of project tasks, and can help you with workspace and process planning. For example, a **TimeLiner** sequence may indicate that when a particular site crane moves from its start point to its end point over the course of a particular afternoon, a workgroup working nearby causes an obstruction along its route. This potential obstruction problem can be resolved before going to site (e.g., the crane can be moved along a different route, the workgroup moved out of the way, or the project schedule altered). See [Add Animation](#) (page 664) for more information.
- Linking **TimeLiner** and **Clash Detective** together enables the time-based clash checks on the project. See [Locate and Manage Interferences](#) (page 669) for further details on time-based clashing.

- Linking **TimeLiner**, Object Animation, and **Clash Detective** together enables clash testing of fully animated **TimeLiner** schedules. So, instead of visually inspecting a **TimeLiner** sequence to make sure, for example, that the moving crane didn't collide with a workgroup, you can run a Clash Detective test.

TimeLiner Window

The **TimeLiner** dockable window enables you to attach items in the model to project tasks, and simulate project schedules.

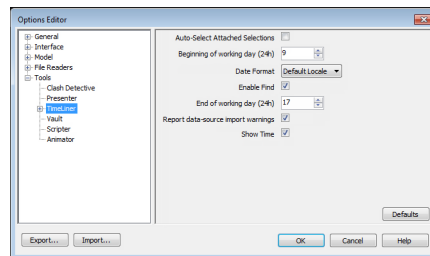
To toggle the TimeLiner window

- Click **Home** tab ➤ **Tools** panel ➤ **TimeLiner** .

 **Menu:** Classic user interface: Tools ➤ TimeLiner

To set the TimeLiner options

- 1 Click  ➤ Options.
- 2 Expand the **Tools** node in the **Options Editor**, and click the **TimeLiner** option.



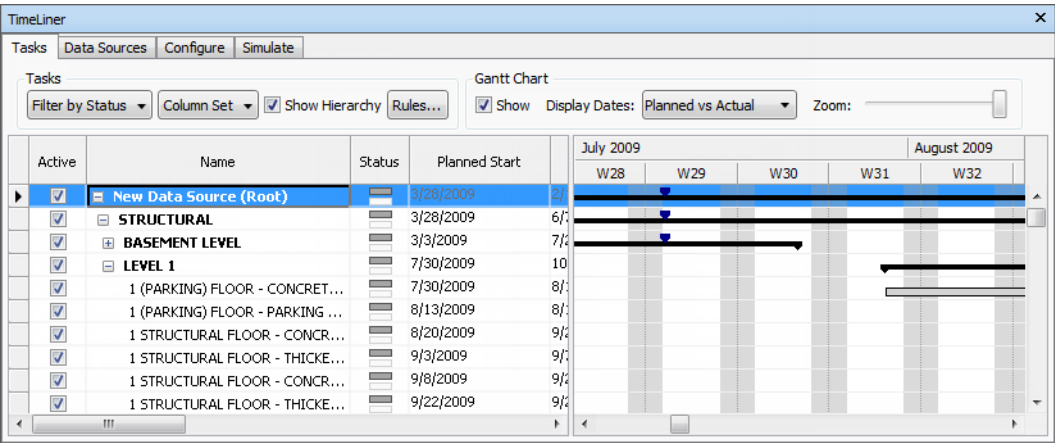
- 3 On the **TimeLiner** page, select the **Auto Select Attached Items** check box if you want **TimeLiner** to select any attached items in the **Scene View** as you select each task in the **TimeLiner** window.
- 4 Select the time that you want the working day to begin using the **Beginning of Working Day (24h)** option.
- 5 Select a date format from the **Date Format** drop-down.

- 6 Select the **Enable Find** check box if you want the find option to be available when you right-click in the **Tasks** tab.
- 7 Select the time that you want the working day to finish using the **End of Working Day (24h)** option.
- 8 Select the **Report Data-Source Import Warnings** check box to enable the displaying of warning messages if an issue is encountered when importing data in the **TimeLiner** window, **Data Sources** tab.
- 9 Select the **Show Time** check box to display the time in the date columns in the **Tasks** tab.

NOTE You can select the format of the text file that will be used to import/export a CSV file into **TimeLiner** in the [TimeLiner CSV Dialog Box](#) (page 823).

Tasks Tab

The **Tasks** tab enables you to create and manage project tasks. It shows all of the tasks in your schedule, listed in a table format. You can use the scroll bars at the right and at the bottom of the tab to move through the task records.



The Task View

The tasks are shown in a multi-column table, which provides some flexibility in how records are displayed. You can:

- Move or resize columns
- Sort column data in ascending or descending order

- Add new user columns to the default column set

TIP You can move between entries with the keyboard. Simply select a task, and use Tab and SHIFT + Tab keyboard keys to move forwards and backwards between fields. The keyboard can then be used to edit and set each entry where necessary.

The Task Hierarchy Autodesk Navisworks**TimeLiner** supports a hierarchical task structure, as imported from a data source, for example Microsoft Project™ (see the [Data Sources Tab](#) (page 613) for more information). The hierarchy can be expanded or contracted by clicking on the plus or minus sign, respectively, to the left of the task.

The Status Icons Each task has its own status identified by an icon. Two separate bars are drawn for each task, showing planned against actual relationships. The color is used to differentiate the early (blue), on-time (green), late (red), and planned (grey) portions of the task. Dots mark the planned start and end dates.

Placing the mouse pointer over a Status icon shows a tooltip explaining the task status.

 **Finished before planned start.**

 **Early start, early finish.**

 **Early start, on-time finish.**

 **Early start, late finish.**

 **On-time start, early finish.**

 **On-time start, on-time finish.**

 **On-time start, late finish.**

 **Late start, early finish.**

 **Late start, on-time finish.**

 **Late start, late finish.**

 **Started after planned finish.**

 **No comparison.**

The Active Check Box The check box in the **Active** column enables you to turn a task on/off. If a task is turned off, then it will not appear in the [simulation](#) (page 617). For hierarchical tasks, turning off the parent task will automatically turn off all child tasks.

The Shortcut Menus Right-clicking within the tasks area on the tab, opens a shortcut menu that enables you to work with tasks in your schedule.

- **Copy Date/Time** - copies date/time values in the selected field. This option is only available when you right-click one of the date fields (for example, **Planned Start**).

NOTE Currently, you cannot use the **CTRL + C** keyboard shortcut to copy date/time values.

- **Paste Date/Time** - pastes date/time values. To access this option you need to right-click one of the date fields. Also, this option is not available unless a valid date/time has been previously copied.

NOTE Currently, you cannot use the **CTRL + V** keyboard shortcut to copy date/time values.

- **Enable Planned Dates** - Simulate Planned Dates for the selected task. This option is available when you right-click the **Planned Start** or the **Planned End** field.
- **Enable Actual Dates** - Simulate Actual Dates for the selected task. This option is available when you right-click the **Actual Start** or the **Actual End** field.
- **Dates** - Enables you to simulate actual and planned dates for the selected task. If no dates are enabled for the task, then it will not appear in the [simulation](#) (page 617).
- **Add Task** - adds a new task to the schedule. This option is available when you right-click in an area of the task view below any current tasks.
- **Attach Current Selection** - attaches the currently selected items in the scene to the selected tasks.
- **Attach Current Search** - attaches all items selected by the current **Search** to the selected tasks. See [Find Objects](#) (page 349) for more information on searching.
- **Attach Set** - attaches all items contained within a Selection Set, to the selected tasks. When you choose this option a list of all Selection and Search Sets saved in the current scene is displayed. Choose the Selection or Search Set you wish to attach to the tasks. See [Create and Use Sets of Objects](#) (page 360) for more information on using Selection and Search Sets.
- **Append Current Selection** - appends the currently selected items in the scene to the items already attached to the selected tasks.

NOTE See [Select Objects](#) (page 337) for more information on how to select items in Autodesk Navisworks.

- **Clear Attachment** - removes the attachment from this task.
- **Add Comment** - adds a comment to the task. See [Use Comments, Red-lines, and Tags](#) (page 399) for more information.
- **Fill Down** - sets the **Task Type** of all currently selected tasks in the Tasks view to match that of the task that is currently 'in focus'.
- **Insert Task** - inserts a new task above the one currently selected in the Task view.
- **Delete Task** - deletes the task currently selected in the Task view.
- **Auto-Add Tasks** - automatically adds a task for every topmost layer, top-most item, or every search and selection set.
- **Find** - finds items in a schedule based on the search criteria that you select in the **Find** menu. This option can be turned on/off in the **Options Editor** (**Tools > TimeLiner > Enable Find** check box).

You can use multi-selection (i.e. holding down SHIFT or CTRL) to perform most commands on several tasks at once. For example, should you need to delete all tasks, select the first task, then hold down SHIFT and select the last task, then press Delete.

Tasks

Column Set The **Column Set** drop-down enables you choose one of three pre-defined column sets to display in the Tasks view; Basic, Standard or Extended. Alternatively you can create a customized column set in the [Choose TimeLiner Columns](#) (page 619) by clicking **Choose Columns**, then selecting **Custom** when you have set up your preferred column set.

Filter by Status The **Filter by Status** drop-down enables you to filter tasks based on their status. Filtering a task temporarily hides the task from the Task and Gantt Chart views, but does not make any changes to the underlying data structure.

Rules Click to display the [TimeLiner Rules Dialog Box](#) (page 619).

Show Hierarchy Clicking the **Show Hierarchy** check box 'flattens' the hierarchical structure shown in the Tasks view.

Gantt Chart

The Display Dates The **Display Dates** drop-down enables you to switch between **Actual**, **Planned**, and **Planned vs Actual** Gantt charts.

The Zoom Slider The **Zoom** slider enables you to adjust the resolution of the displayed Gantt chart. The utmost left position selects the smallest available increment in the timeline (for example, days); the utmost right position selects the largest available increment in the timeline (for example, years).

Show Click the **Show** check box to show or hide the Gantt chart.

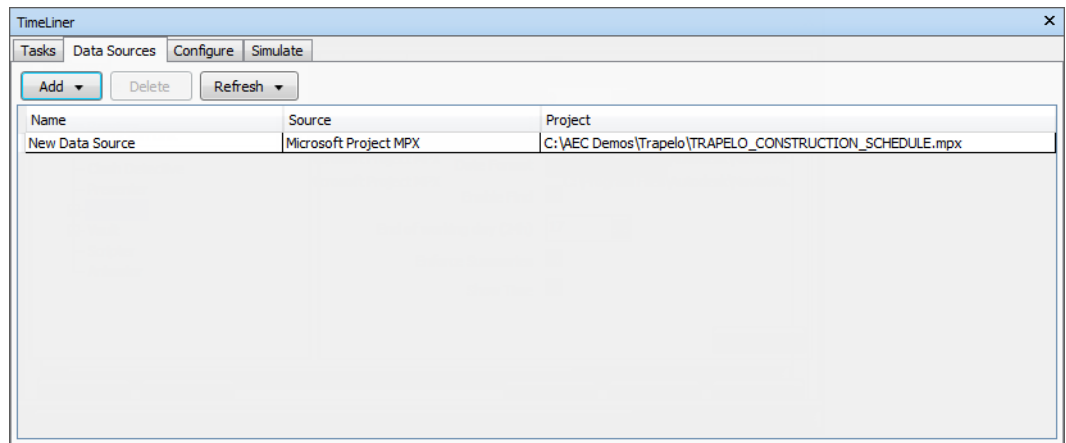
The Gantt Chart View

The Gantt Chart displays a colored bar chart illustrating your project status. Each task takes up one row. The horizontal axis represents the time span of the project, broken down into increments (such as days, weeks, months, and years) and the vertical axis represents the project tasks. Tasks can run sequentially, in parallel, or overlapping.

You can drag a task to different dates, or click and drag on either end of the task to extend or shorten its duration. Any changes are automatically updated in the Task view.

Data Sources Tab

The **Data Sources** tab enables you to import tasks from a third-party scheduling software, such as Microsoft Project, Asta, and Primavera. It shows all added data sources, listed in a table format.



The Data Source View

The data sources are shown in a multi-column table. The columns show name, source (for example, Microsoft Project™) and project (e.g. *my_schedule.mpp*). Any further columns (there may be none) identify the fields from the external schedule which specify the task type, unique id, start date and end date for each imported task.

You can move and resize columns, if necessary.

The Data Source Buttons

Add Creates a new connection to an external project file. Clicking this button displays a menu listing all project sources that may be connected to on the current machine. See [Supported Scheduling Software](#) (page 651) for more information on which sources are available.

Delete Deletes the currently selected data source. If you have refreshed the data source before you deleted it, any tasks and data read from the data source will remain on the **Tasks** tab.

Refresh Displays the [Refresh from Data Source Dialog Box](#) (page 624) where you can refresh the selected data sources.

The Shortcut Menus

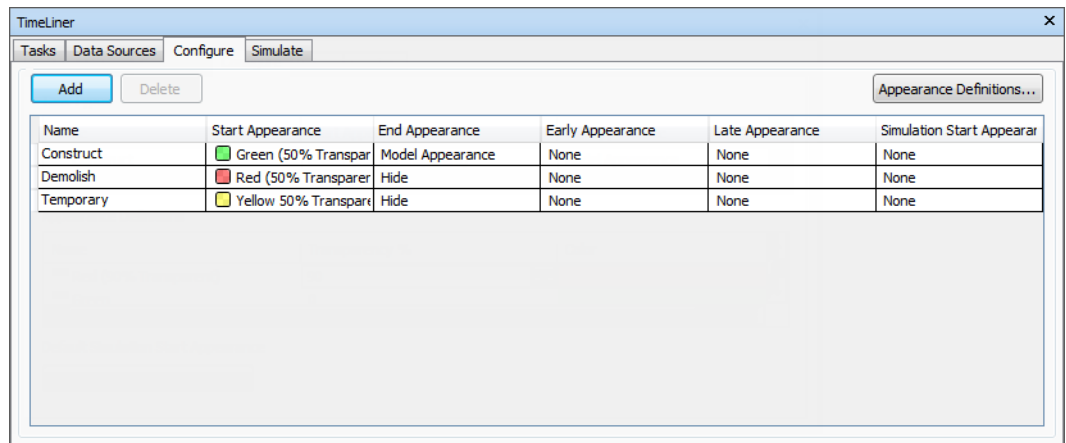
Right-clicking within the data source area on the tab opens a shortcut menu that enables you to manage data sources.

- **Rebuild Task Hierarchy** - reads all of the tasks and associated data (as defined in the Field Selector Dialog Box) from the selected data sources and adds these to the **Tasks** tab. Choose this option also to synchronize with the selected project file when new tasks have been added to the project file. This will rebuild the task hierarchy in **TimeLiner**, containing all of the latest tasks and data.
- **Synchronize** - Updates all existing tasks in the **Tasks** tab, with the latest associated data from the selected data sources (for example, Start and End dates).
- **Delete** - deletes the currently selected data source. If you have refreshed the data source before you deleted it, any tasks and data read from the data source will remain on the **Tasks** tab.
- **Edit** - enables you to edit the selected data source. This will display the [Field Selector Dialog Box](#) (page 620) , from which you can define new fields or re-define existing ones.
- **Refresh** - displays the [Refresh from Data Source Dialog Box](#) (page 624) where you can refresh the selected data source.
- **Rename** - enables you to rename the data source to something more appropriate. When the text field becomes highlighted, enter the new name, then press Enter to save it.

NOTE Tasks in the data source will be ignored if they do not include both start and end dates, with the start date being less than or equal to the end date.

Configure Tab

The **Configure** tab allows you to set up the task parameters, such as task types, appearance definitions for tasks, and the default model appearance at the start of the simulation.



Task Types

The tasks types are shown in a multi-column table. You can move and resize table columns, if necessary.

NOTE You can double-click on the **Name** column to rename a task type, or double-click on any other column to change the task type appearance.

TimeLiner comes with three predefined task types:

- **Construct** - for tasks where the attached items are to be constructed. By default, during a simulation, the objects are highlighted in green at the start of the task and are reset to [Model Appearance](#) (page 661) at the end of the task.
- **Demolish** - for tasks where the attached items are to be demolished. By default, during a simulation, the objects are highlighted in red at the start of the task and are hidden at the end of the task.
- **Temporary** - for tasks where the attached items are only temporary. By default, during a simulation, the objects are highlighted in yellow at the start of the task and are hidden at the end of the task.

Add

Adds a new task type.

Delete

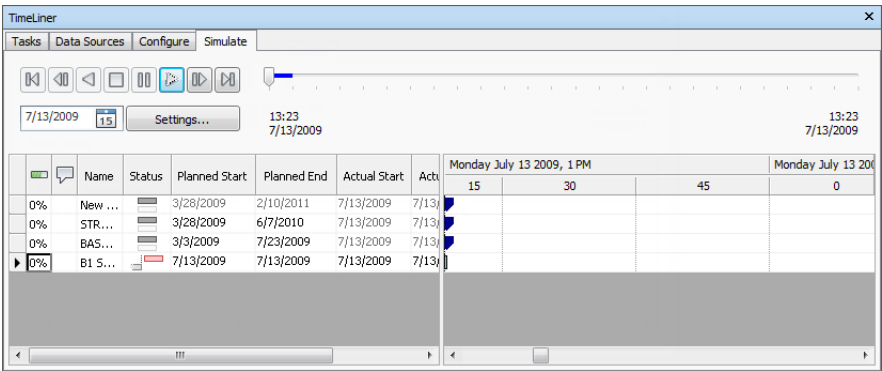
Deletes the selected task type.

Appearance Definitions

Opens the [Appearance Definitions Dialog Box](#) (page 632) where you can set up and change appearance definitions.

Simulate Tab

The **Simulate** tab enables you to simulate your **TimeLiner** sequence throughout the duration of the project schedule.



The Playback Controls

Use the standard VCR buttons to step and play forwards and backwards through the simulation:

Rewind  will rewind the simulation back to the beginning.

Step Back  will step back a single step size. See [Simulation Settings Dialog Box](#) (page 624) for more information.

Reverse Play  will play the simulation backwards.

Pause  will pause the simulation at the time you press it at. You can then look around and interrogate the model, or step forwards and backwards through the simulation. To continue playing from where you paused, just press **Play** again.

Stop  will stop the simulation playing and rewind back to the beginning.

Play  will play the simulation from the currently selected time.

Step Forwards  will step forwards a single step size.

Forward  will fast forward the simulation to the end.

You can use the **Simulation Position** slider to quickly move forwards and backwards through the simulation. Full left is at the beginning and full right is at the end.



The **Date/Time** box next to the VCR buttons shows the point in time through the simulation. You can click on the drop-down icon to the right of the date to display a calendar, from which you can select a date to 'jump' to.

The Settings Button

The **Settings** button opens the [Simulation Settings Dialog Box](#) (page 624) that enables you to define how the schedule is simulated.

The Task View

All active tasks are shown in a multi-column table. You can move and resize table columns, if necessary.

You can view the current simulation time for each of the active tasks, and how close to completion they are (**Progress** is displayed as a percentage). The **Status** of each active task is also displayed as an icon. For simulations where **Planned** and **Actual** dates are available, the status provides a visual representation as to whether there is any variance between the planned and actual dates. See [The Status Icons](#) (page 609) for more information.

The Gantt Chart View

The Gantt Chart displays a colored bar chart illustrating your project status. Each task takes up one row. The horizontal axis represents the time span of the project, broken down into increments (such as days, weeks, months, and years) and the vertical axis represents the project tasks. Tasks can run sequentially, in parallel, or overlapping.

The visible range (zoom) level is determined by the **Interval Size** options in the [Simulation Settings dialog](#) (page 624).

Choose TimeLiner Columns Dialog Box

Use this dialog box to customize the columns display in the **TimeLiner Tasks** tab. To access it, click the **Column Set** drop-down ► **Choose Columns** button on the **Tasks** tab.

Column List Displays all the available columns. Click a column check box to select it for display in the **Tasks** tab.

Move Up Moves the selected column(s) up the list by one place.

Move Down Moves the selected column(s) down the list by one place.

Show All Selects all check boxes in the list.

Hide All Clears all check boxes in the list.

TimeLiner Rules Dialog Box

Use the **TimeLiner Rules** dialog box to create and manage **TimeLiner** rules.

Click the **Rules** button on the **Tasks** tab to open this dialog box. It lists all currently available rules. These can be used to map tasks to items in the model. Each of the default rules can be edited and new rules may be added as necessary.

New Opens the **Rules Editor** dialog box where you can create a new rule.

Edit Opens the **Rules Editor** dialog box where you can edit the currently selected rule.

Delete Deletes the currently selected rule.

Import/Export Attachment Rules  Enables you to import/export rules to/from an XML file.

NOTE Imported rules will overwrite any existing rules with the same name.

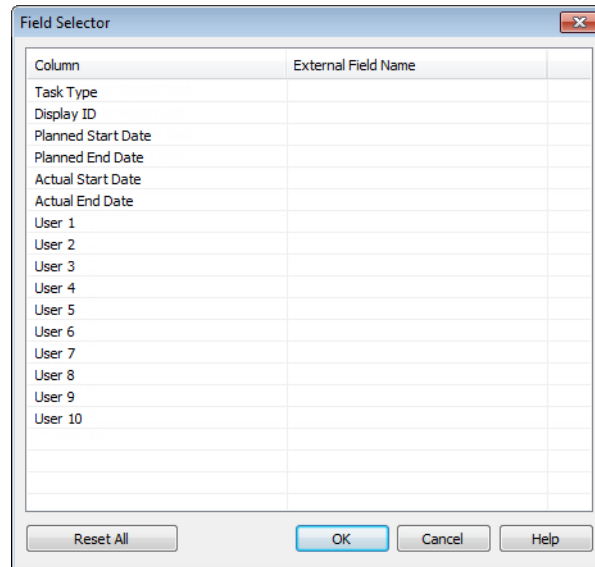
Override Current Selections check box If selected, then when the rules are applied they will replace any existing attached items. If not selected, the rules will attach items to the relevant tasks without attached items.

Apply Rules Applies the selected rules.

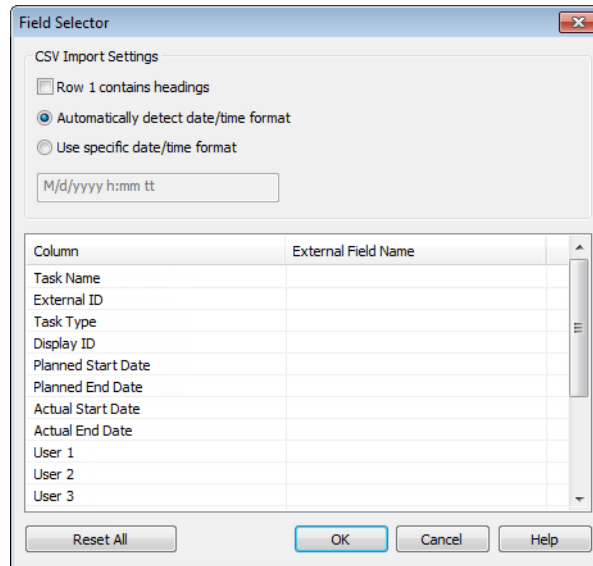
Field Selector Dialog Box

The **Field Selector** dialog box determines various options used when importing data from an external project schedule.

The options available may be different for each type of data source.



The Field Selector Dialog Box for importing data from an external scheduling software.



The Field Selector Dialog Box for importing CSV data

Options

CSV Import Settings

Row 1 Contains Headings Select the **Row 1 Contains Headings** check box if you want the first row of data in your CSV file to be treated as column headings. **TimeLiner** will use it to populate the **External Field Name** options in the grid.

If the first row of data in your CSV file does not contain column headings, clear this check box.

Automatically Detect Date/Time Format Select the **Automatically Detect Date/Time Format** option if you want **TimeLiner** to attempt to determine the date/time format used in your CSV file. First, TimeLiner applies a set of rules to try to establish the date/time format used in the document; if it is not possible, it uses the local settings on your system.

Use Specific Date/Time Format Select the **Use Specific Date/Time Format** option if you want to manually specify the date/time format that should be used. When this radio button is selected, you can enter the required format into the box provided. See the list of valid date/time codes below.

NOTE If one or more date/time-based columns are found to contain fields where the data cannot be mapped to a valid date/time value using the manually-specified format, **TimeLiner** will 'fall back' and attempt to use the automatic date/time format.

The Field-Mapping Grid

The field-mapping grid is a grid containing, in the left column, all the columns from the current **TimeLiner** schedule and, in the right column, a number of drop-down menus enabling you to map the incoming fields to the **TimeLiner** columns.

NOTE When you are importing data from a CSV file, the **External Field Name** column of the grid shows the data from the first row of the CSV file if **Row 1 Contains Headings** is selected. Otherwise, it defaults to **Column A, Column B**, and so on.

Task Name This field is shown and required when you import CSV data. If you do not map this field, you will receive an error message.

External ID The field is used to uniquely identify each imported task. This allows synchronization to work even if major changes are made to the external schedule in the scheduling software. The default behavior is to use the most appropriate field for each source. Some sources do not have a well defined unique id, in which case you may need to choose a field manually.

NOTE It is recommended that you have a column in the CSV file containing unique data, for example an incrementing number, and map it to this field. If this field is not mapped then you will be unable to rebuild or synchronize the data source link.

Task Type The field is used to automatically assign [Task Types](#) (page 615) to each imported task.

Display ID The field is used for any additional user requirements and does not need to be mapped.

Planned Start Date The field is used to identify a planned start date. This allows planned against actual comparisons to be made and simulated.

Planned End Date The field is used to identify a planned end date. This allows planned against actual comparisons to be made and simulated.

Actual Start Date Some project sources support multiple start dates for different purposes. The default behavior is to use the most appropriate available date for each source. This field may be used to define an **Actual Start** date, should it be different to that selected by default.

Actual End Date Some project sources support multiple end dates for different purposes. The default behavior is to use the most appropriate available date for each source. This field may be used to specifically define an **Actual End** date, should it be different to that selected by default.

User 1 to 10 Ten user fields are available to link any custom data fields from the project source.

Reset All button Use this button to clear all column mapping and also reset CSV import settings to their defaults, if appropriate.

Valid Date/Time Codes

d, %d The day of the month. Single-digit days will NOT have a leading zero.

dd The day of the month. Single-digit days will have a leading zero.

ddd The abbreviated day name.

dddd The full day name.

M, %M The numeric month. Single-digit months will NOT have a leading zero.

MM The numeric month. Single-digit month names will have a leading zero.

MMM The abbreviated month name.

MMMM The full month name.

y, %y The year without the century. If less than 10, it will NOT have a leading zero.

yy The year without the century. If less than 10, it will have a leading zero.

yyyy The year in four digits, including the century.

h, %h The hour in 12-hour clock format. Single digit hours will NOT have a leading zero.

hh The hour in 12-hour clock format. Single digit hours will have a leading zero.

H The hour in 24-hour clock format. Single digit hours will NOT have a leading zero.

HH The hour in 24-hour clock format. Single digit hour will have a leading zero.

m, %m The minute. Single-digit minutes will NOT have a leading zero.

mm The minute. Single-digit minutes will have a leading zero.

s, %s The second. Single-digit seconds will NOT have a leading zero.

ss The second. Single-digit seconds will have a leading zero.

t, %t The first character of the AM/PM designator, if any.

tt The AM/PM designator, if any.

z The GMT time zone offset (“+” or “-” followed by the hour only). Single-digit hours will NOT have a leading zero.

zz The time zone offset. Single digit hours will have a leading zero.

zzz The full time zone offset in hours and minutes. Single digit hours and minutes will have leading zeros. For example, “-8:00”.

Refresh from Data Source Dialog Box

Use this dialog box to choose how data is refreshed.

Rebuild Reads all of the tasks and associated data (as defined in the [Field Selector Dialog Box](#) (page 620)) from the selected external schedule(s) and adds these to the **Tasks** tab. Choose this option also to synchronize with the selected external schedule when new tasks have been added to the project file. This will rebuild the task hierarchy in **TimeLiner**, containing all of the latest tasks and data.

Synchronize Updates all existing tasks in the **Tasks** tab, with the latest associated data from the selected external schedule(s) (e.g. Start and End dates).

Pointing device: **TimeLiner Data Sources** tab ► **Refresh**

Simulation Settings Dialog Box

The **Settings** button on the **Simulate** tab provides access to the **Simulation Settings** dialog box.

Simulation Settings

☐ **Override Start / End Dates**

Start Date
8:00:00 AM 4/ 1/2009

End Date
5:00:00 PM 2/10/2011

Interval Size
5 Percent

☐ **Show all tasks in interval**

Playback Duration (Seconds)
30

Overlay Text
Edit Top

Animation
No Link

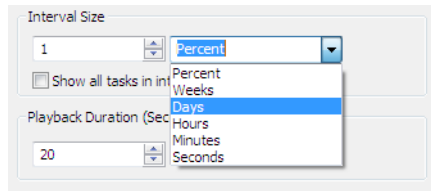
View
☐ Planned
☐ Planned (Actual Differences)
☐ Planned against Actual
☒ Actual
☐ Actual (Planned Differences)

OK Cancel Help

It is possible to override the **Start** and **End** dates that the simulation runs between. Selecting the **Override Start/End Dates** check box enables the date boxes and allows you to choose the start and end dates. By doing this, you can simulate a small sub-section of the overall project. The dates will be shown on the **Simulate** tab. These dates will also be used when exporting animations.

You can define the **Interval Size** to use when stepping through the simulation using the playback controls. The interval size can be set either as a percentage of the overall simulation duration or to an absolute number of days or weeks, and so on.

Use the drop-down list to select the interval unit, then use the Up and Down arrow buttons to increase or decrease the interval size.

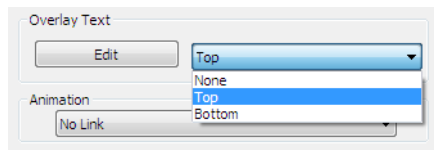


It is also possible to highlight all the tasks that are being worked on during the interval. By selecting the **Show All Tasks in Interval** check box, and, for example, setting the **Interval Size** to 5 Days, all tasks being worked on during those 5 days will be set to their **Start Appearance** in the **Scene View**, including those that begin and end within the bounds of the interval. The **Simulation** slider will show this by drawing a blue line under the slider. If this check box is clear, tasks that begin and end within the bounds of the interval will not be highlighted in this manner, and will need to overlap with the current date in order to be highlighted in the **Scene View**.

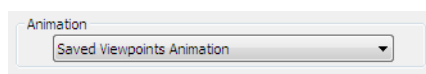
You can define the overall **Playback Duration** for the complete simulation (the time needed to play it through from start to finish). Use the **Up** and **Down** arrow buttons to increase or decrease the duration (in seconds). You may also enter a duration directly into this field.

You can define whether the current simulation date should be overlaid in the **Scene View**, and if so whether it should appear at the top or bottom of the screen. From the drop-down, choose from **None** (to display no overlay text), **Top** (to display the text at the top of the window), or **Bottom** (to display the text at the bottom of the window).

You can **Edit** the information displayed in the overlay text using the [Overlay Text Dialog Box](#) (page 630). This dialog box also makes it possible to alter the Font **Type**, **Style** and **Size** by clicking on the contained **Font** button.



You can add animation to an entire schedule, so that during the **TimeLiner** sequence playback, Autodesk Navisworks will also play the specified viewpoint animation or camera.



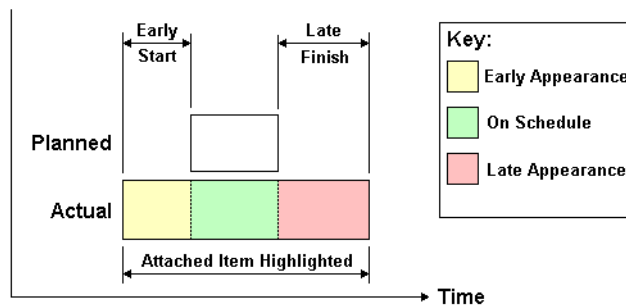
The following options can be selected in the **Animation** field:

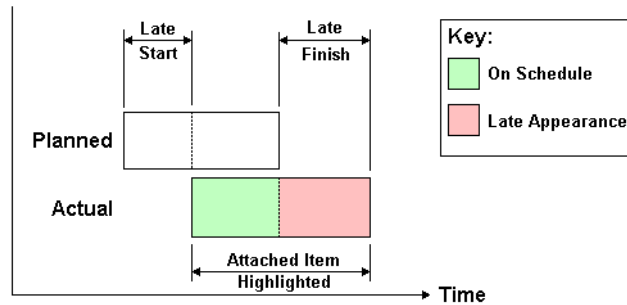
- **No Link** - no viewpoint animation or camera animation will be played.
- **Saved Viewpoints Animation** - links your schedule to the currently selected viewpoint or viewpoint animation.
- Scene X - ➤ **Camera** - links your schedule to a camera animation in the selected animation scene.

You can pre-record suitable animations for use with the **TimeLiner** simulation (see [Record and Play Animations](#) (page 471)). Using animation also affects the [Animation Export](#) (page 494).

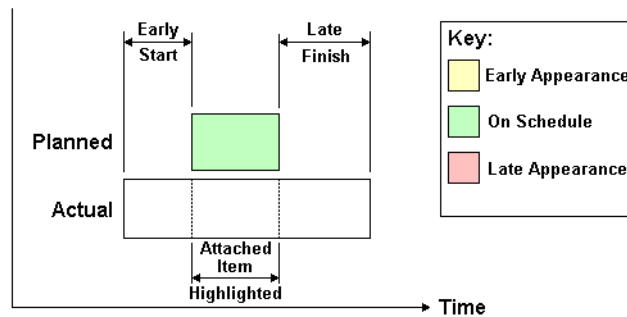
View area. Each view will playback the schedule depicting **Planned** and **Actual** relationships:

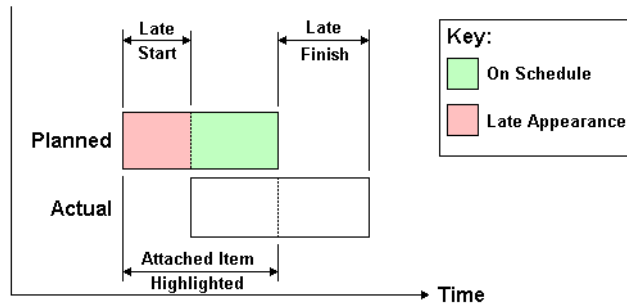
- **Actual**. Choose this view to simulate the **Actual** schedule only (that is, only use the **Actual Start** and **Actual End** dates).
- **Actual (Planned Differences)**. Choose this view to simulate the **Actual** schedule against the **Planned** schedule. This view will only highlight the items attached to the task over the **Actual** date range (that is, between **Actual Start** and **Actual End**. See diagram below for graphical representation). For time periods where the **Actual** dates are within the **Planned** dates (on schedule), the items attached to the task will be displayed in the **Task Type Start Appearance**. For time periods where the **Actual** dates are early, or late in comparison to the **Planned** dates (there is a variance), then the items attached to the task will be displayed in the **Task TypeEarly** or **Late Appearance**, respectively.



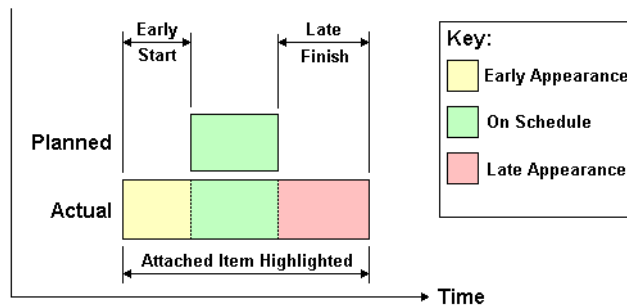


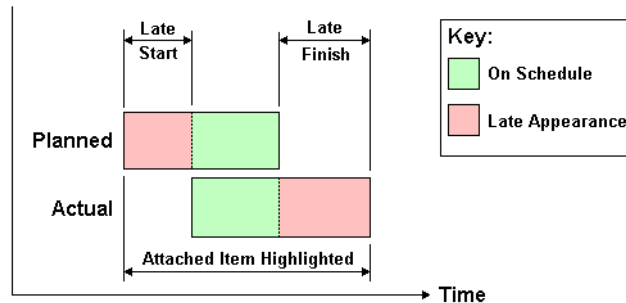
- **Planned.** Choose this view to simulate the **Planned** schedule only (that is, only use the **Planned Start** and **Planned End** dates).
- **Planned (Actual Differences).** Choose this view to simulate the **Actual** schedule against the **Planned** schedule. This view will only highlight the items attached to the task over the **Planned** date range (that is, between **PlannedStart** and **Planned End**. See diagram below for graphical representation). For time periods where the **Actual** dates are within the **Planned** dates (on schedule), the items attached to the task will be displayed in the **Task TypeStart Appearance**. For time periods where the **Actual** dates are early, or late in comparison to the **Planned** dates (there is a variance), then the items attached to the task will be displayed in the **Task TypeEarly** or **Late Appearance**, respectively.





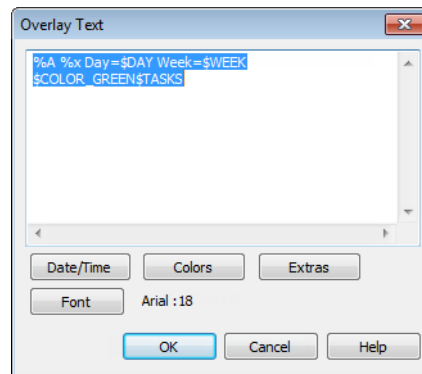
- **Planned Against Actual.** Choose this view to simulate the **Actual** schedule against the **Planned** schedule. This will highlight the items attached to the task over the entire **Planned** and **Actual** date range (that is, between the earliest of **Actual** and **PlannedStart** dates and the latest of **Actual** and **PlannedEnd** dates. See diagrams below for graphical representation). For time periods where the **Actual** dates are within the **Planned** dates (on schedule), the items attached to the task will be displayed in the **Task TypeStart Appearance**. For time periods where the **Actual** dates are early, or late in comparison to the **Planned** dates (there is a variance), then the items attached to the task will be displayed in the **Task TypeEarly** or **Late Appearance**, respectively.





Overlay Text Dialog Box

You have the option to define the text overlaid in the **Scene View** during simulation, by clicking **Edit** in the [Simulation Settings Dialog Box](#) (page 624).



By default the date and time are displayed using the format specified in **Control Panel ► Regional Settings**. You can specify the exact format to use by entering text into the text box. Most text will appear as entered, except that words prefixed with a “%” or “\$” character act as keywords and are replaced with various values. The **Date/Time** and **Extras** buttons can be used to select and insert all possible keywords. The **Colors** button can be used to define the color of the overlay text.

The **Font** button brings up the standard Microsoft Windows font picker dialog box. Once the correct font, font style and point size have been selected, press **OK** to return to the **Overlay Text** dialog box. The current font selection is

shown next to the **Font** button, and during the **TimeLiner** simulation, all text on the overlay will be shown using this font.

Date/Time Keywords

%a Abbreviated weekday name.

%A Full weekday name.

%b Abbreviated month name.

%B Full month name.

%c Date and time representation appropriate for locale.

%d Day of month as decimal number (01 - 31).

%H Hour in 24-hour format (00 - 23).

%I Hour in 12-hour format (01 - 12).

%j Day of year as decimal number (001 - 366).

%m Month as decimal number (01 - 12).

%M Minute as decimal number (00 - 59).

%p Current locale's A.M./P.M. indicator for 12-hour clock.

%S Second as decimal number (00 - 59).

%U Week of year as decimal number, with Sunday as first day of week (00 - 53).

%w Weekday as decimal number (0 - 6; Sunday is 0).

%W Week of year as decimal number, with Monday as first day of week (00 - 53).

%x Date representation for current locale.

%X Time representation for current locale.

%y Year without century, as decimal number (00 - 99).

%Y Year with century, as decimal number.

%z Time-zone abbreviation; no characters if time zone is unknown.

%Z Time-zone name; no characters if time zone is unknown.

Color Keywords

\$COLOR_RED Sets the overlay display text color to be red.

\$COLOR_BLUE Sets the overlay display text color to be blue.

\$COLOR_GREEN Sets the overlay display text color to be green.

\$COLOR_WHITE Sets the overlay display text color to be white.

\$COLOR_BLACK Sets the overlay display text color to be black.

\$RGBr,g,b\$RGB Sets the overlay display text to any color specified using explicit RGB values between 0 and 255. For example, “\$RGB127,127,127\$RGB” sets the color to grey.

Extra Keywords

\$TASKS Adds the name of each currently active task to the overlay display text. Each task is displayed on a new line.

\$DAY Days since start of first task in project (starting from 1).

\$WEEK Weeks since start of first task in project (starting from 1).

CTRL + Enter Type CTRL + Enter to insert a new line into the overlay display text.

%% Percent sign.

Appearance Definitions Dialog Box

Use this dialog box to customize the default task types, or create new ones, as necessary. To access it, click the **Appearance Definitions** button on the **Configure** tab.

TimeLiner comes with a set of ten predefined appearance definitions that you can use to configure the task types. Appearances define a level of transparency and a color.

Name Specifies the appearance definition name. Click on the name to change it as required.

Color Specifies the appearance definition color. Click on the color to change it as required.

Transparency Specifies the appearance definition transparency. Use the slider or enter a value to change the transparency as required.

Add Click to add an appearance definition.

Delete Click to delete the currently selected appearance definition.

Default Simulation Start Appearance This drop-down box specifies a default appearance to apply to all objects in the model at the start of the simulation. The default is **Hide**, which is ideal for simulating most construction sequences.

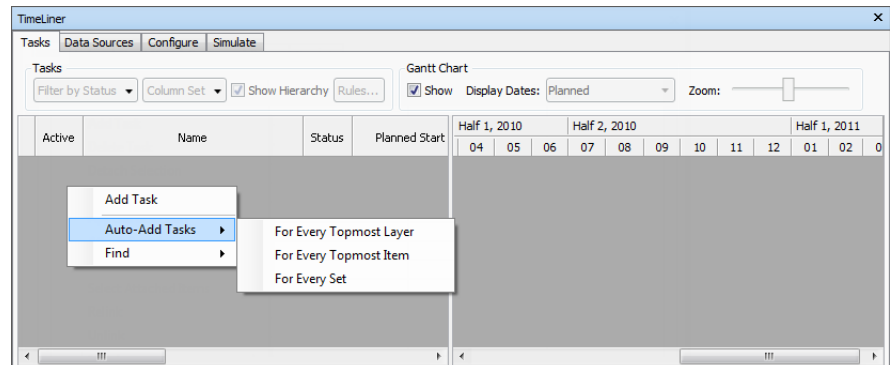
Get Started

To get started

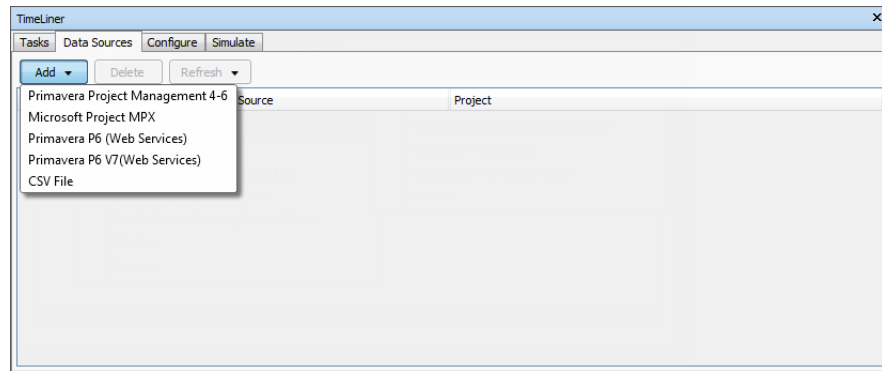
- 1 Load a model into Autodesk Navisworks in the usual way (see [Open Files](#) (page 209) for more information), and then click **Home** tab ► **Tools** panel ► **TimeLiner** .

NOTE Although the ability to create 4D simulations is limited to users with access to the **TimeLiner** functionality, any user can actually play 4D simulations back once they have been set up.

- 2 Create some tasks, each having a name, start and end date and a task type. You can [add tasks manually](#) (page 639). Alternatively, right-click within the task area on the **Tasks** tab to open the [Tasks Tab](#) (page 609), and create an initial set of tasks based on layer, item, or selection set names. **TimeLiner** defines some default task types for you (Construct, Demolish and Temporary), or you can define your own task types on the [Configure Tab](#) (page 615). You can also view a read-only graphical representation of your project schedule, and can switch between **Planned**, **Actual**, and **Planned vs Actual** Gantt charts.



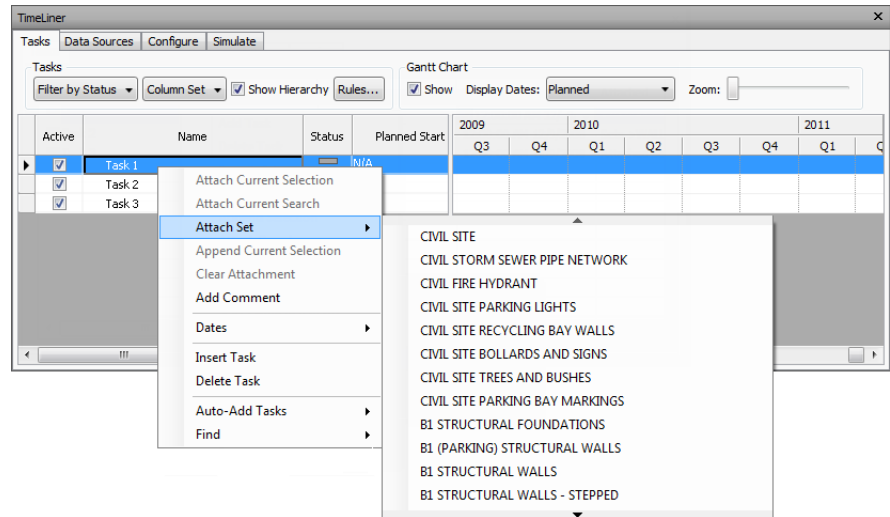
You can use the [Data Sources Tab](#) (page 613) to import tasks from an external source, such as Microsoft Project™. You can choose a field from the external schedule to define the types of the imported tasks, or you can set task types by hand. You can edit task parameters directly in **TimeLiner**. You can also update the schedule externally and **refresh** the tasks in **TimeLiner** to align them with those from the external source. Any changes you make to tasks imported from external project files will be overwritten next time you refresh the corresponding data sources.



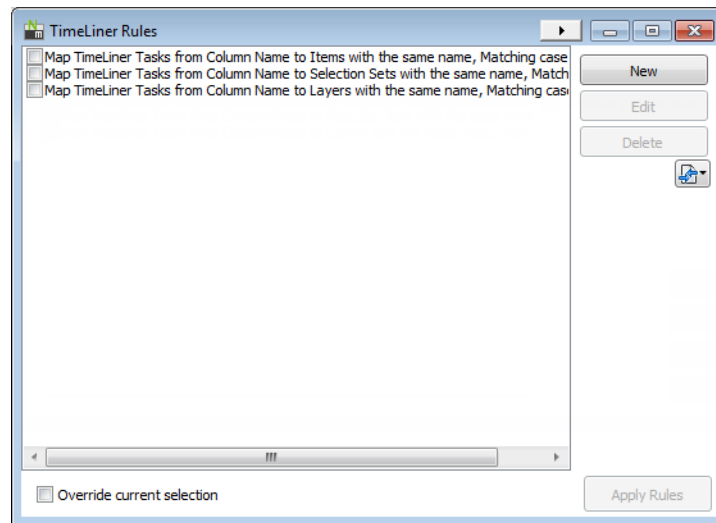
3 Attach objects in the model to tasks.

If you used the shortcut menu on the [Tasks Tab](#) (page 609) to create an initial set of tasks based on layer, item or selection set names, then the corresponding layers, items or selection sets will already be attached for you.

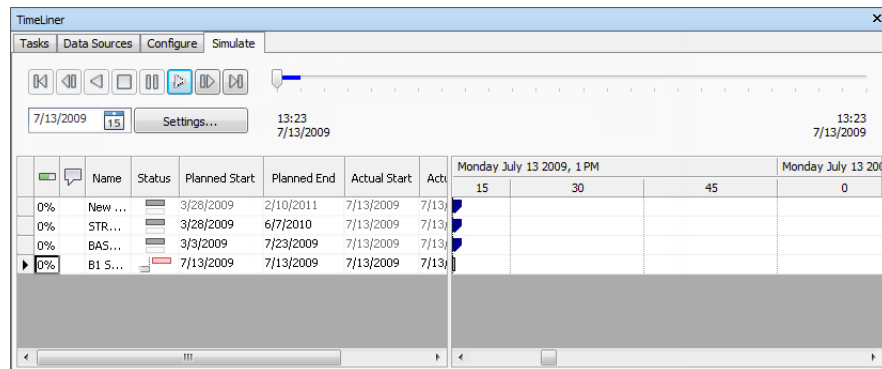
If you need to manually [Attach Tasks to Geometry](#) (page 644), you can use the shortcut menu to attach selections, searches or selection sets.



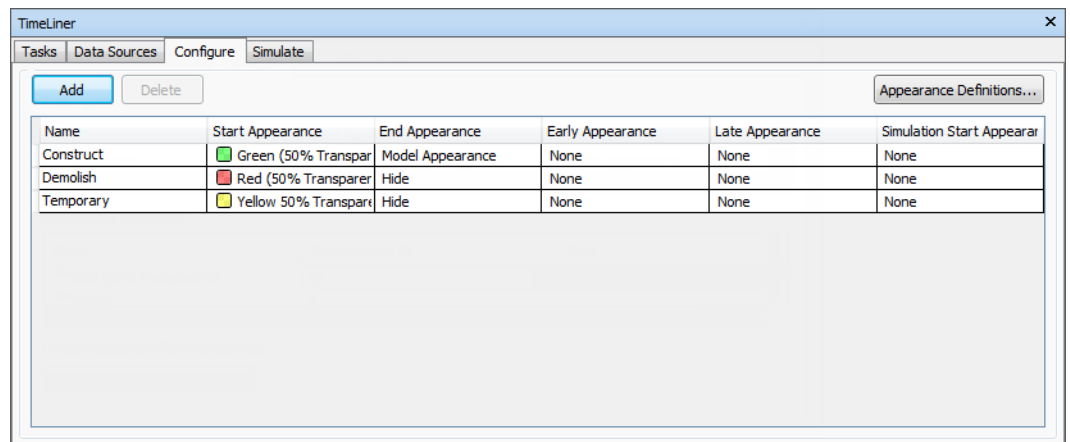
Alternatively, you can [Use Rules to Attach Tasks](#) (page 646) automatically.



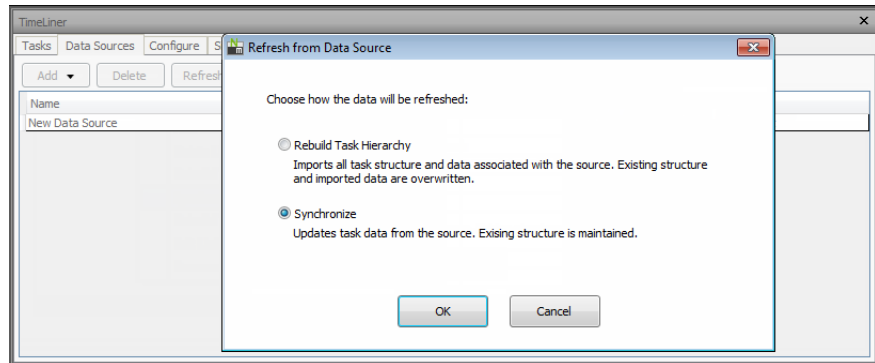
- 4 [Simulate your schedule](#) (page 660). You can visualize your model at any date in your schedule with the currently active tasks highlighted. Run through the entire schedule using familiar VCR controls. You can [Add Animation](#) (page 664) to your construction schedules, and enhance the quality of simulations.



- 5 [Adjust the way your simulation plays](#) (page 661). You can use the **Configure** tab to create new task types and edit old ones. The task type defines what happens at the start and end of each task of that type. You can hide attached objects, change their appearance or reset them to the appearance specified in the model.



- 6 Create image and AVI files. [Export](#) (page 664) the current simulation view as an image, or the entire simulation as an AVI.
- 7 Keep up to date with changes in your project. Save your project as a Autodesk Navisworks NWF file. Open the NWF and [Synchronize Tasks with Project Changes](#) (page 659).



TimeLiner Tasks

The **Tasks** tab can be used to create and edit tasks, to attach tasks to geometry items, and to validate your project schedule.

You can adjust the Task View.

You can also add new user columns to the default column set. This is useful when you import data from external project files that contain more fields than **TimeLiner**.

To move a column

- 1 Open the **TimeLiner** window, and click the **Tasks** tab.
- 2 Position the mouse over the header of the column you want to move, and press the left mouse button.
- 3 Drag the column header to the desired location, and release the left mouse button.

To resize a column

- 1 Open the **TimeLiner** window, and click the **Tasks** tab.
- 2 Select the grid line at the right border of the column header of the column you want to resize.
- 3 Drag to the right to enlarge the column, or to the left to shrink it.

To specify sort order

- 1 Click the header of the column you want to use, and click again to alternate between ascending/descending.

Ascending order sorts the column starting with the smallest value and ending with the largest value (for example: a-z, 0-9, Monday-Friday).

Descending order sorts the column starting with the largest value and ending with the smallest value.

NOTE If sorting is selected on the **Status** column, activities will be sorted from early through to late start for ascending, and vice-versa for descending. The task hierarchy is preserved during the sorting process, that is tasks are sorted first by container, then by container contents.

NOTE You can only sort by one column at a time. Instead of using the shortcut menu to change the sort order, you can click the heading of the desired column. This alternates the sort order between ascending and descending.

To add a user column

- 1 In the **TimeLiner** window, the **Tasks** tab, click the **Column Set** drop-down ► **Choose Columns** button.
- 2 In the **Choose TimeLiner Columns** dialog box, select the check box next to one of the ten available user columns, for example **User 1**, and click **OK**.

The custom user column has been added to **TimeLiner**. By default, it's got the same name as the option you selected in the dialog box.

- 3 Right-click the added column, click **Rename User Column**, and type in a new name, for example 'Cost'.

NOTE To populate this column with data, you must map the user field (in this case, User 1) to the corresponding data field in each of the data sources. For more information, see [Field Selector Dialog Box](#) (page 620).

To delete a user column

- 1 In the **TimeLiner** window, the **Tasks** tab, click the **Column Set** drop-down ► **Choose Columns** button.
- 2 In the **Choose TimeLiner Columns** dialog box, clear the check box next to the user column you want to remove, and click **OK**.

Create Tasks

In **TimeLiner** tasks can be created in one of the following ways:

- Manually, one at a time.
- Automatically, based on object structure in the **Selection Tree**, or the selection and search sets.
- Automatically, built from data sources added to **TimeLiner**.

TIP Unlike manual tasks, which need to be attached to geometry in your model, automatic tasks will be attached to the corresponding geometry as soon as they are created.

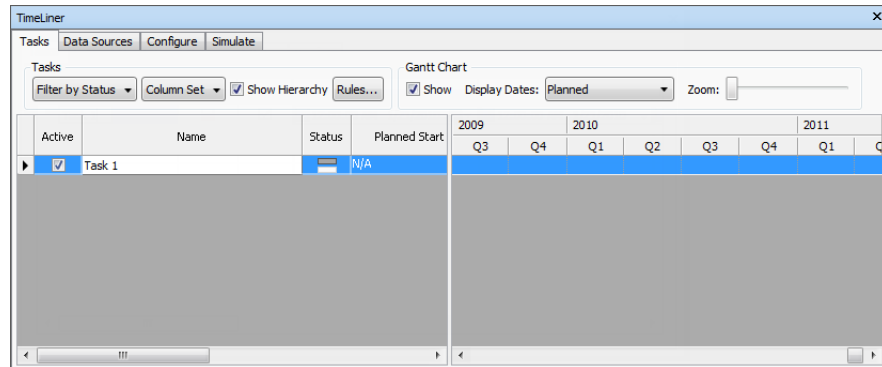
To add a task manually

- 1 Load your model into Autodesk Navisworks (see [Open Files](#) (page 209) if you need help).
- 2 Click **Home** tab ► **Tools** panel ► **TimeLiner** , and click the **Tasks** tab in the **TimeLiner** window.
- 3 Right-click anywhere in the task view, and click **Add Task** on the shortcut menu.

NOTE You can click on an existing task and select **Insert Task** to insert a task above the selected task.

- 4 Enter the name for your task, and press Enter. The task is now added to your schedule.

NOTE If you press Enter when the bottom task in the task view is selected, then a new task will be created below it.



To add tasks based on the Selection Tree Structure

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window, the **Tasks** tab, right-click in the Task View, and click **Auto-Add Tasks** on the shortcut menu.
- 3 Click **For Every Topmost Layer** if you want to create tasks with the same names as each topmost layer in the **Selection Tree**.
Click **For Every Topmost Item** if you want to create tasks with the same names as each topmost item in the **Selection Tree**. This can be a layer, a group, block or cell, or geometry, depending on how the model is constructed.

NOTE **PlannedStart** and **End** dates will be automatically created, starting from the current system date and incrementing by one day for each subsequent end and start date. The **Task Type** will be set to **Construct**.

To add tasks based on search or selection sets

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window, the **Tasks** tab, right-click in the Task View, and click **Auto-Add Tasks** on the shortcut menu.
- 3 Click **For Every Set** to create tasks with the same names as each selection and search set in the **Sets** dockable window.

NOTE **PlannedStart** and **End** dates will be automatically created, starting from the current system date and incrementing by one day for each subsequent end and start date. The **Task Type** will be set to **Construct**.

Edit Tasks

You can edit any of the task parameters directly in **TimeLiner**. However, the changes you make to tasks imported from external project files will be overwritten next time you refresh the corresponding data sources.

To change the task name

- 1 In the **TimeLiner** window, the **Tasks** tab, select the row with the task you want to modify, and click its name.
- 2 Type in the new name for the task, and press Enter.

 **Command entry:** F2

To change the task date and time

NOTE By default, time is not shown. To show time for tasks, open the **Options Editor**, click **Tools > TimeLiner**, and select the **Show Time** check box.

- 1 In the **TimeLiner** window, the **Tasks** tab, click the task you want to modify.
- 2 Modify the task's date:

Clicking the drop-down button in the **Actual Start** and **Actual End** field opens the calendar, and lets you set the **ActualStart/End** dates.

Clicking the drop-down button in the **Planned Start** and **Planned End** field opens the calendar, and lets you set the **PlannedStart / End** dates.



Use the left and right arrow buttons at the top of the calendar to move backwards and forwards a month, respectively, then click on the day you require.

- 3 To change the start or end time, click the time unit you want to modify (hours, minutes or seconds), and enter the value. You can use the left and right arrow keys to move between the units in the time field.

14-Feb-11 9:33 AM ▼

To copy and paste date/time values

- 1 In the **TimeLiner** window, the **Tasks** tab, right-click the date field you want to copy, for example **Planned Start**.
- 2 Click **Copy Date/Time**.
- 3 Right-click in a different date field, and click **Paste Date/Time**.

NOTE Currently, you cannot use the keyboard shortcuts **CTRL + C** and **CTRL + V** to copy and paste date/time values.

To set or modify the task type

- 1 In the **TimeLiner** window, the **Tasks** tab, click the task you want to modify.
- 2 From the **Task Type** drop-down list, select the type of task you wish this task to be. By default, you can choose from:
 - Construct
 - Demolish
 - Temporary

The task type defines how the items attached to the task will be displayed during simulation. For example, a default construction sequence would start with all items hidden, as the task starts the attached items will be displayed in transparent green, then as the task ends the attached items will be displayed as they are in the normal model display (this may be with materials applied if previously set up in the **Presenter** tool). Task types themselves can be defined and new types created on the [Configure Tab](#) (page 615).

NOTE You can quickly change the task type for all selected tasks. To do this, press SHIFT or CTRL key and select required tasks, right-click the selection, and click **Fill Down** on the shortcut menu. The task type specified for the first selected task will be used for all tasks in the range.

To delete a task

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window, the **Task** tab, right-click the task you want to delete, and click **Delete Task** on the shortcut menu.

NOTE Deleting the root-level task deletes all tasks in the hierarchy. If you made a mistake, use the **Undo**  button on the **Quick Access** toolbar to restore the deleted item.

Use Gantt Charts

The **Gantt Chart** view in the **Tasks** tab provides a visual representation of your tasks. The horizontal axis represents the time span of the project, broken down into increments (such as days, weeks, months, and years) and the vertical axis represents the project tasks. Tasks can run sequentially, in parallel, or overlapping.

You can drag a task to different dates, or click and drag on either end of the task to extend or shorten its duration. Any changes are automatically updated in the **Task** view. Similarly, modifying a field in the **Tasks** view modifies the corresponding field on the **Gantt Chart** view.

To view a Gantt chart for Actual dates

- 1 Click **Home** tab ► **Tools** panel ► **TimeLiner** , and click the **Tasks** tab in the **TimeLiner** window.
- 2 Select **Actual** in the **Display Dates** drop-down.

To view a Gantt chart for Planned dates

- 1 Click **Home** tab ► **Tools** panel ► **TimeLiner** , and click the **Tasks** tab in the **TimeLiner** window.

- 2 Select **Planned** in the **Display Dates** drop-down.

To view a Gantt chart for Planned vs Actual dates

- 1 Click **Home** tab ► **Tools** panel ► **TimeLiner** , and click the **Tasks** tab in the **TimeLiner** window.
- 2 Select **Planned vs Actual** in the **Display Dates** drop-down.

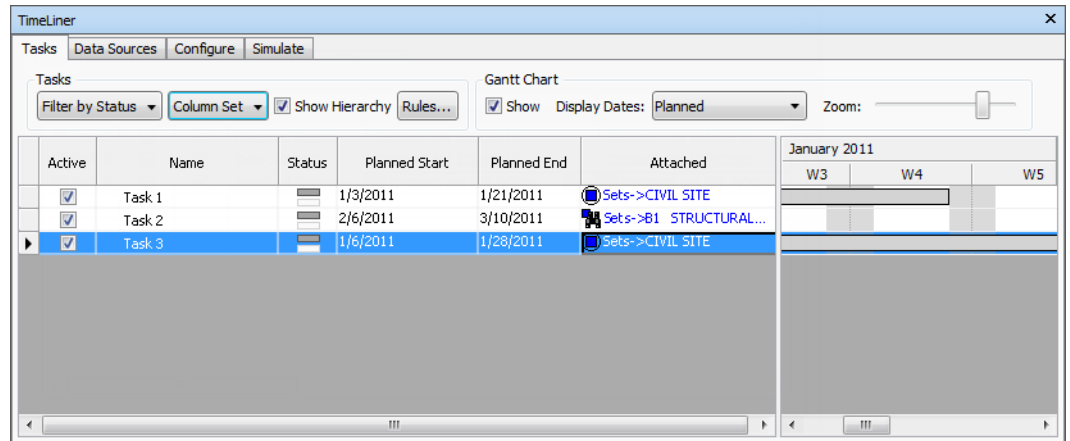
To change a resolution of a Gantt chart

- 1 Click the **Tasks** tab in the **TimeLiner** window.
- 2 Optional: Use the **Display Dates** drop-down to customize the displayed Gantt chart. By default, **Planned** dates are used.
- 3 Drag the **Zoom** slider to adjust the chart's resolution. The utmost left position selects the smallest available increment in the timeline (for example, days); the utmost right position selects the largest available increment in the timeline (for example, years).

Attach Tasks to Geometry

Each task needs to be attached to items in the model before 4D simulation can work. You can create and attach tasks at the same time, or you can create all tasks first, and then attach them, either separately, or in rule-defined batches.

Tasks can be attached to the current selection in the **Scene View**, or any of the selection sets, or any of the search sets. You can view the type of attachments in the **Attached** column. By default, this column is located after **Task Type**, but you can move it, for example, in front of the **Start** column.



Attach Tasks Manually

To attach a task to a current selection

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Select the desired geometry objects in the **Scene View**, or on the **Selection Tree**.
- 3 In the **TimeLiner** window, the **Tasks** tab, right-click the required task, and click **Attach Current Selection** on the shortcut menu.

To attach a task to a selection set or a search set

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 In the **TimeLiner** window, the **Tasks** tab, right-click the required task, click **Attach Set**, and click the required selection set or a search set.

To attach a task to a current search

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Run the desired search from the **Find Items** dockable window.

- 3 In the **TimeLiner** window, the **Tasks** tab, right-click the required task, and click **Attach Current Search**.

To attach multiple tasks to a selection set or a search set

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window, the **Tasks** tab, hold the CTRL or SHIFT key to select all required tasks.
- 3 Right-click the task selection, click **Attach Set**, and click the required selection set or a search set.

To attach multiple tasks to a current selection

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 Select the desired geometry objects in the **Scene View**, or on the **Selection Tree**.
- 3 In the **TimeLiner** window, the **Tasks** tab, hold the CTRL or SHIFT key to select all required tasks.
- 4 Right-click the task selection, and click **Attach Current Selection** on the shortcut menu.

TIP If you need to add additional items to an already attached task, click the **Append Current Selection** option on the shortcut menu, otherwise any previous task attachments will be overridden. If you made a mistake, right-click the task, and click **Clear Attachment**.

Use Rules to Attach Tasks

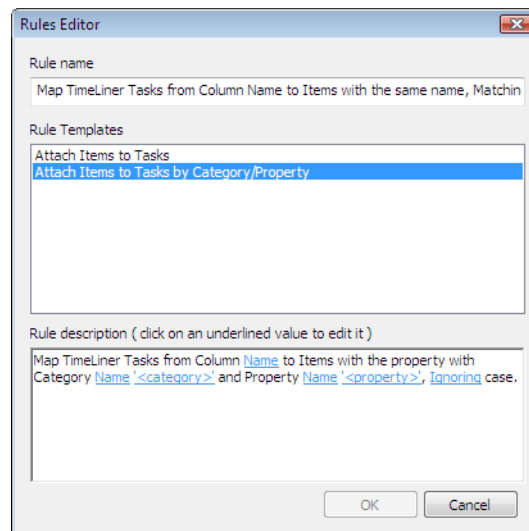
Attaching tasks manually could take a long time. It is a good practice to either use task names, which correspond to **Selection Tree** layers, or create selection and search sets that correspond to the task names. If this is the case, then you can apply predefined and custom-made rules to instantly attach tasks to objects in the model.

Predefined Rules

- **Items By Task Name.** Choose this rule to attach each geometry item in the model to each task with the same name in the specified column. The default is to use the **Name** column.
- **Selection Sets By Task Name.** Choose this rule to attach each **Selection** and **Search Set** in the model to each task with the same name in the specified column. The default is to use the **Name** column.
- **Layers By Task Name.** Choose this rule to attach each layer in the model to each task with the same name in the specified column. The default is to use the **Name** column.
- **Attach Items to Task by Category/Property.** Choose this rule to attach each item in the model with the defined property, to each task with the same name in the specified column. The default is to use the **Name** column; however, you will also need to define the **<category>** and **<property>** values.

To add custom TimeLiner rules

- 1 From the **Tasks** tab of the **TimeLiner** window, click the **Rules** button.
- 2 In the **TimeLiner Rules** dialog box, click the **New** button.
The **Rules Editor** dialog is displayed.



- 3 Enter a new name for your rule in the **Rule Name** box.

NOTE If you choose not to enter a name, upon selecting a rule template the name of that template will be used.

- 4 From the **Rule Templates** list, choose a template from which your rule will be based upon.

NOTE The **Attach Items to Tasks** template is the one used for the first three pre-defined **TimeLiner** rules, that is, **Items**, **Selection Sets** and **Layers By Task Name**.

The **Attach Items to Tasks by Category/Property** template enables you to specify a property within the model scene. If a task has the same name as the specified property value in the model, then all items having that property will be attached to that task when you check the rule **Attach Items to Tasks by Category/Property**, and click **Apply Rules**.

- 5 In the **Rule Description** box, click on each of the **underlined values** to define your custom rule. The customizable values available with the built in templates are:
 - **Column Name.** Choose which column in the [Tasks Tab](#) (page 609) you want to compare **Item** names to. The default is the **Name** column, though you may also choose other available columns, for example one of the ten **User** columns, identified in the [Field Selector Dialog Box](#) (page 620).
 - **Items.** Choose what you would like the value in the column to be compared to in the model scene. The default is **Item** name, though you may also choose either **Selection Sets** or **Layers**.
 - **Matching.** Use **case sensitivity** and therefore only match names that are exactly the same. You may also choose **Ignoring** to ignore case sensitivity.
 - **Category/Property Name.** Use the name of the category or property as it is displayed in the interface (recommended). You can also choose **Internal Name** which is that accessed via the API (for advanced use only).
 - **"<category>".** Choose from the available list, which category the property you wish to define is in. Only the categories that are contained in the scene are available in the drop down.
 - **"<property>".** Choose from the available list, which property you wish to define. Again, only the properties in the scene within the chosen category will be available.

- 6 Click **OK** to add the new **TimeLiner** rule, or **Cancel** to return to the **TimeLiner Rules** dialog box.

To apply TimeLiner rules

- 1 Open the **TimeLiner** window, and click the **Rules** button on the **Tasks** tab.
- 2 Select the check boxes for all rules you want to apply. The rules will be applied in order.

NOTE You can also click **Import/Export Attachment Rules** button  which enables you to import/export rules to/from an XML file. Imported rules will overwrite any current rules with the same name.

- 3 If **Override Current Selections** check box is selected, then when the rules are applied they will replace any existing attached items. Otherwise, the rules will attach items to the relevant tasks without attached items.
- 4 Click **Apply Rules**.

NOTE To validate tasks in your schedule, click the **Tasks** tab, right-click in the **Task View**, and use the **Find** shortcut menu. This option can be turned on/off in the **Options Editor** (**Tools** > **TimeLiner** > **Enable Find** check box).

Validate Project Schedule

You can check the validity of your schedule, by identifying items that have not been included in any task, are duplicated in multiple tasks or, are in overlapping tasks. This is especially useful, when you use rules to attach tasks.

An item may remain unattached for a number of reasons. For example, a task in the project scheduling file is omitted, or the geometry item has not been included in a selection or search set.

To check a schedule

- 1 If the **TimeLiner** window is not already open, click **Home** tab > **Tools** panel > **TimeLiner** .
- 2 In the **TimeLiner** window, the **Tasks** tab, right-click the tasks view, and click **Find**. If this option is not available, open the **Options Editor**, and click **Tools** > **TimeLiner** > **Enable Find** check box.

3 Select one of the available options:

- **Unattached/Uncontained Items.** Choose this to select any items in the scene that are not attached to a task, or are not contained within any other item attached to a task.

NOTE A **Contained** item is the child of another item. For example, if you select a **Group, Block** or **Cell** which is comprised of various pieces of geometry, then that geometry is contained within the **Group, Block** or **Cell**. A **Group, Block** or **Cell** may therefore be attached to a task and although the child geometry is not directly attached itself, it is contained within an item that is attached.

- **Contained Items.** Choose this to select any items in the scene that are either attached to a task or contained within any other items attached to a task.
- **Attached Items.** Choose this to select any items in the scene that are directly attached to a task.
- **Items Attached to Multiple Tasks.** Choose this to select any items in the scene that are directly attached to more than one task.
- **Items Contained in Multiple Tasks.** Choose this to select any items in the scene that are either attached to, or contained within any other item that is attached to more than one task.
- **Items Attached to Overlapping Tasks.** Choose this to select any items in the scene that are attached to more than one task, where the task durations overlap.
- **Items Contained in Overlapping Tasks.** Choose this to select any items in the scene that are either attached to, or contained within any other item that is attached to more than one task, where the task durations overlap.

The check results are highlighted on the **Selection Tree** and in the **Scene View**.

Link to External Project Files

One of **TimeLiner** most powerful features is its integration with project scheduling software. A list of tasks including their start and end dates/times can be imported from a project file directly into **TimeLiner**.

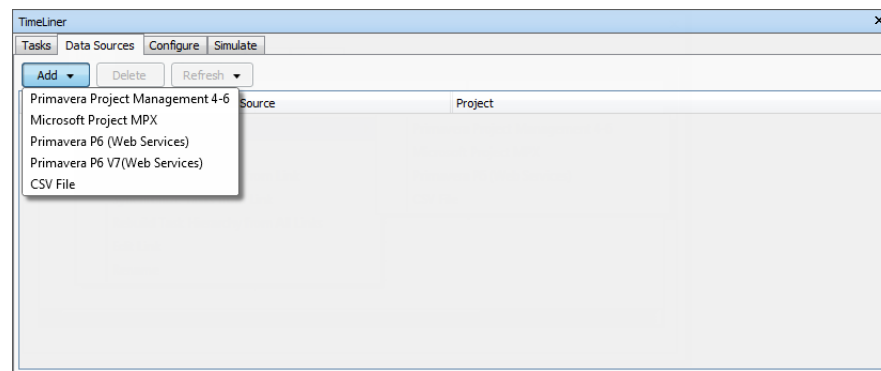
NOTE When opening files in Autodesk Navisworks 2012 that were created in Autodesk Navisworks 2011 or previous, you may find that the **Planned** and **Actual** dates have been swapped. Autodesk Navisworks now uses **Planned** dates by default instead of **Actual** dates, as is consistent with most planning applications.

To switch date sets for affected files, edit the data source (see [Edit a Data Source](#) (page 658)), switch the mapping of the **Planned** and **Actual** dates in the [Field Selector Dialog Box](#) (page 620), then rebuild the task hierarchy (see [Build Tasks from Data Sources](#) (page 659)).

Supported Scheduling Software

Autodesk Navisworks**TimeLiner** supports a variety of scheduling software.

NOTE Some of these may only function if the relevant scheduling software is installed.



NOTE Autodesk Navisworks**TimeLiner** supports multiple sources of external schedules using a COM interface. Anyone can develop support for a new schedule source - the type library that defines the interface is included with **TimeLiner**.

Microsoft Project 2003

This data source requires Microsoft Project 2003™ to be installed.

Microsoft Project 2007

This data source requires Microsoft Project 2007™ to be installed.

Microsoft Project MPX

TimeLiner can read Microsoft Project™ MPX files directly, without needing a copy of Microsoft Project™ (or any other scheduling software) installed. Primavera SureTrak™, Primavera Project Planner and Asta Power Project™ can all export MPX files.

NOTE Primavera SureTrak™ exports its unique id in the text10 field of the MPX file, rather than the unique id field. When linking to an MPX file exported from SureTrak be sure to specify the text10 field as the unique id field in the [Field Selector Dialog Box](#) (page 620) .

Primavera Project Planner (P3)

This data source requires Primavera Project Planner™ to be installed. Once the link is added, its status will be displayed as “Status not available” in the TimeLiner.

NOTE TimeLiner will ask for a user name and password to link to Primavera Project Planner. If these are unknown, there is a possibility that these can be retrieved from the P3 shortcut. To locate this information, right-click on the shortcut used to run P3, and select **Properties**. The **Properties** dialog box will show the target command line which will look something like

```
"C:\Program  
Files\P3\P3.exe /U:Bob"
```

Here, the user name is set as “Bob”, and for the purposes of the **TimeLiner** tool, the password will be the same.

Primavera Project Management 4.1, 5.0, and 6.2

This data source requires a number of elements to be installed alongside Autodesk Navisworks:

- Primavera Project Manager 4.1, or 5.0, or 6.2 product
- ActiveX Data Objects 2.1
- Primavera Software Development Kit (available on the Primavera CDs)

PPM 4 - 6 are database driven, and as such require the Software Development Kit to be installed in order to set up the ODBC data source link. This can be

installed and setup from the Project Management CD by taking the following steps:

- 1 Insert Project Management CD, enter the Product Key and accept the **License Agreement**.
- 2 Make sure **Primavera Applications or Components** is selected, and click **Next**.
- 3 Select **Other Components**, and click **Next**.
- 4 Select **Software Development Kit**, and click **Next**.
- 5 Continue through clicking **Next** until the install starts.
- 6 Once installation is complete, click **OK** to start the **Database Configuration** wizard.
- 7 Adjust settings in the **Software Development Kit Setup** dialog box where appropriate, and click **OK**.
- 8 Click **Yes** for a log file, followed by **Finish** to complete.

On connecting to PPM 4 - 6 within **TimeLiner**, a logon dialog box allows the source link to be selected (a warning occurs if none are present). The user name and password are not stored in the Autodesk Navisworks file, and will be prompted for each time.

Once connected, a dialog box allows the user to select which project to open. A check box determines if all sub-projects are opened.

By default, **TimeLiner** brings through **Start, End, Planned Start**, and **Planned End** as **Start, Finish, BL Start** and **BL Finish** respectively.

Other Primavera dates can be selected through the generic [Field Selector Dialog Box](#) (page 620), including **Project Activity Codes, Global Activity Codes**, and **User Defined Fields**.

The **TimeLiner** hierarchy supports the WBS structure for Project / Activity hierarchy.

NOTE Due to the nature of the Primavera Project Manager 4 - 6 products using the SDK for data access, importing data with **TimeLiner** can take a longer period of time than other formats.

Primavera P6 (Web Services)

Accessing the Primavera P6 Web Services capability greatly speeds up the time taken to synchronize **TimeLiner** and Primavera schedules.

This data source requires you to set up a Primavera Web Server. Refer to the Primavera P6 Web Server Administrator Guide (available in your Primavera documentation).

Asta Power Project 8 - 10

This data source requires the corresponding version of Asta Power Project™ to be installed.

CSV Support

The **TimeLiner** tool supports importing and exporting schedules in CSV/Excel formats.

You need to use commas (",") when saving a CSV file, otherwise you will have problems importing it into **TimeLiner**.

NOTE If your software product uses other separators than commas (","), try adjusting the software settings. You may also want to check your OS setting for the regional list separator.

You can use the **Options Editor** to change the format of the text file that will be used to import/export a CSV file into **TimeLiner**.

See also:

[Import Data from an External Project Schedule](#) (page 654)

[Export TimeLiner CSV](#) (page 500)

Add and Manage Data Sources

In this section you will learn how to create, delete and edit data sources.

Import Data from an External Project Schedule

To import data from an external project schedule

- 1 If the **TimeLiner** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **TimeLiner**  .

- 2 In the **TimeLiner** window click the **Data Sources** tab.
- 3 Click the **Add** button and choose the required option from a list of the project sources that may be connected to on the current PC.

NOTE For more information on which sources are typically available, see [Supported Scheduling Software](#) (page 651).

- 4 Use the standard **Open** dialog box to browse to and open the desired project file.

TimeLiner accesses the selected file in accordance with the predefined parameters configured for the corresponding data source using a COM interface.

Depending on the data source, the [Field Selector Dialog Box](#) (page 620) may be displayed. You can use it to override some of the predefined data import options.

- 5 By default, your data source is called “New Data Source(x)”, where “x” is the latest available number. To make your data source more descriptive, right-click it, click **Rename** on the shortcut menu, and enter a new name.

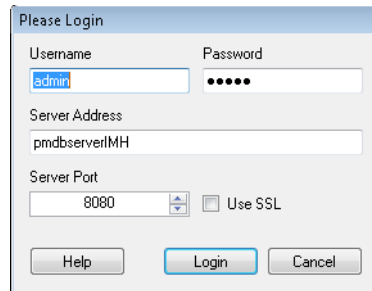
To import data from a Primavera P6 project

NOTE Follow the same procedure to import data from a Primavera P6 V7 project, selecting the Primavera P6 V7 options where applicable.

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window, click the **Data Sources** tab.
- 3 Click the **Add** button and click **Primavera P6 (Web Services)**.

NOTE If you can’t see this option, you must set up a Primavera Web Server first. Refer to the Primavera P6 Web Server Administrator Guide (available in your Primavera documentation).

- 4 When the Primavera login dialog box is displayed, enter your user name, password, and the server address.



The image shows a 'Please Login' dialog box with the following fields and controls:

- Username:** A text field containing 'admin'.
- Password:** A text field with masked characters (dots).
- Server Address:** A text field containing 'pmdbservetlMH'.
- Server Port:** A spin box set to '8080'.
- Use SSL:** An unchecked checkbox.
- Buttons:** 'Help', 'Login', and 'Cancel' buttons at the bottom.

TIP The **Server Address** is the name of a machine on your domain or an IP address for the server.

- 5 In the **Primavera P6 Database Instance Selection** dialog box double-click the desired **Instance ID** to select it.
- 6 In the **Primavera P6 Project Selection** dialog box double-click the desired project file to open it.
- 7 Use the [Field Selector Dialog Box](#) (page 620) to override some of the predefined data import options.
TimeLiner connects to the selected project file.

To import CSV data

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 In the **TimeLiner** window click the **Data Sources** tab.
- 3 Click the **Add** button and click **CSV File**.
- 4 Use the standard **Open** dialog box to locate the desired project file in CSV format, and click **Open**.
- 5 In the [Field Selector Dialog Box](#) (page 620), use the **CSV Import Settings** area to specify how your data should be imported into **TimeLiner**.

NOTE The CSV import settings, including field mappings, are remembered by the system and pre-populated when the dialog box opens. If you are linking to a different CSV file than was previously used, **TimeLiner** will attempt to map any columns to similarly-named columns in the CSV file.

It is recommended that you have a column in the CSV file containing unique data, for example an incrementing number, and map this to the **External ID** column. This allows data synchronization to work even if major changes are made to the CSV file. If this field is not mapped then you will be unable to rebuild or synchronize the data source.

- Select the **Row 1 Contains Headings** check box if you want the first row of data in your CSV file to be treated as column headings. **TimeLiner** will use it to populate the **External Field Name** options in the grid.

If the first row of data in your CSV file does not contain column headings, clear this check box.

- Select the **Automatically Detect Date and Time** option if you want **TimeLiner** to attempt to determine the date/time format used in your CSV file. First, **TimeLiner** applies a set of rules to try to establish the date/time format used in the document; if it is not possible, it uses the local settings on your system.

Select the **Use Specific Date/Time Format** option if you want to manually specify the date/time format that should be used. When this radio button is selected, you can enter the required format into the box provided. For the list of valid codes, see [Field Selector Dialog Box](#) (page 620).

NOTE If one or more date/time-based columns are found to contain fields where the data cannot be mapped to a valid date/time value using the manually-specified format, **TimeLiner** will 'fall back' and attempt to use the automatic date/time format.

- 6 By default, your data source is called "New Data Source(x)", where "x" is the latest available number. To make your data source more descriptive, right-click it, click **Rename** on the shortcut menu, and enter a new name.

You are now ready to [Build Tasks from Data Sources](#) (page 659).

Edit a Data Source

To edit a data source

- 1 If the **TimeLiner** window is not already open, click **Home** tab **Tools** **TimeLiner**  .
- 2 In the **TimeLiner** window, the **Data Sources** tab, right-click the data source you want to modify, and click **Edit** on the shortcut menu.
The [Field Selector Dialog Box](#) (page 620) opens.
Typically, this dialog box is used to customize the column mapping between the **TimeLiner** and the external project file. For example, if the external file includes **BaselineStart** and **End** dates you may want to map them to the **Start Date** and **End Date** columns in **TimeLiner**. Also, if you have added any [user columns](#) (page 637) to **TimeLiner**, for each link, you must map the data in the external file to the required user columns in **TimeLiner**, otherwise the user columns will be empty.
- 3 Click **OK**.

NOTE For each data source, you must either rebuild the task hierarchy or synchronize tasks to apply the changes.

Delete a Data Source

To delete a data source

- 1 If the **TimeLiner** window is not already open, click **Home** tab **Tools** panel **TimeLiner**  .
- 2 In the **TimeLiner** window, the **Data Sources** tab, right-click the row with data source you want to delete, and click **Delete** on the shortcut menu.
Any tasks and data that have been read from the data source will remain on the **Tasks** tab.

NOTE If you made a mistake, use the Undo  button on the **Quick Access** toolbar to restore the deleted item.

Build Tasks from Data Sources

To build tasks from a data source

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 In the **TimeLiner** window, the **Data Sources** tab, click the **Refresh** button, select either **Selected Data Source** or **All Data Sources** then select **Rebuild Task Hierarchy** in the [Refresh from Data Source Dialog Box](#) (page 624) and click **OK**.
This will import all of the tasks from the project file into **TimeLiner**.
NOTE You can also right-click on a file in the **Data Sources** tab and select the **Rebuild Task Hierarchy** option.
- 3 Click the **Tasks** tab to view the created tasks. The task table is populated in accordance with predefined settings for the data source. You can make any necessary overrides in the [Field Selector Dialog Box](#) (page 620).

NOTE Although the tasks have now been imported into **TimeLiner**, you still need to [Attach Tasks to Geometry](#) (page 644) before you can run a 4D simulation. The fastest way to attach imported tasks is by applying rules (see [Use Rules to Attach Tasks](#) (page 646)).

Synchronize Tasks with Project Changes

Any changes to tasks and task data in the external data source can be easily updated in **TimeLiner**.

To refresh tasks to reflect project changes

- 1 Make changes to the project in the scheduling software, and save it.
- 2 Load your model into Autodesk Navisworks (see [Open Files](#) (page 209) if you need help).
- 3 Click **Home** tab ► **Tools** panel ► **TimeLiner**  , and click the **Data Sources** tab in the **TimeLiner** window.
- 4 Click the **Refresh** button and select **Selected Data Sources** or **All Data Sources**.

- 5 In the [Refresh from Data Source Dialog Box](#) (page 624) select one of the following options and click **OK**.
 - **Rebuild Task Hierarchy**. Choose this to re-import all tasks and associated data from the selected project file(s), and rebuild the task hierarchy on the **Tasks** tab.
 - **Synchronize**. Choose this to update all existing tasks in the **Tasks** tab, with the latest associated data from the selected project file(s) (for example, **Start** and **End** dates).

TIP You can also right-click data source in the **Data Sources** tab and select the **Synchronize** or **Rebuild Task Hierarchy** option.

4D Simulation

In this section, you will learn how to play 4D simulations, and how to customize the simulation playback and appearance.

Play Simulations

To play a simulation

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 On the **Tasks** tab select the **Active** check box for all tasks that you want to include in the simulation.
- 3 Be sure the active tasks are assigned the correct [Task Types](#) (page 615).
- 4 Be sure the active tasks are [attached](#) (page 644) to geometry objects, and click the **Simulate** tab.
- 5 Click the **Play**  button.

The **TimeLiner** window displays the tasks as they are carried out, and the **Scene View** shows the sections of the model added or removed over time, in accordance with the task types.

NOTE When the **Simulate** tab is in use, by default the current simulation date is overlaid in the **Scene View**. You can customize the displayed information by using the [Overlay Text Dialog Box](#) (page 630).

Configure Simulations

You can adjust both the simulation playback parameters, and the simulation appearance.

Simulation Playback

By default, the playback duration of the simulation is set to 20 seconds, regardless of the duration of the tasks. You can adjust the simulation duration, and a number of other playback options to increase the effectiveness of your simulation.

To adjust a simulation playback

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Click the **Simulation** tab, and click the **Settings** button.
- 3 When the [Simulation Settings Dialog Box](#) (page 624) opens, modify the playback settings, and click **OK**.

Simulation Appearance

Each task has a task type associated with it, which specifies how the items attached to the task are treated (and displayed) at the start and end of the task during simulation. The available options are:

- **None** - the items attached to the task will not change.
- **Hide** - the items attached to the task will be hidden.
- **Model Appearance** - the items attached to the task will be displayed as they are defined in the model. This may be the original CAD colors or, if you have applied color and transparency overrides in Autodesk Navisworks or materials in **Presenter**, then these will be displayed.
- **Appearance Definitions** - enables you to choose from the list of **Appearance Definitions**, including ten predefined appearances and any custom appearances you have added.

To add a task type definition

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 Click the **Configure** tab, and click the **Add** button.
- 3 A new task type is added to the bottom of the list; it will be highlighted, enabling you to enter a new name for it.
- 4 Select one of the **Appearance** fields, and click it to open a drop-down menu and assign the desired object behavior.

You can customize the following properties:

- **Start Appearance** - how the items will look at the start of the task, for example Transparent Green.
- **End Appearance** - how the items will look when the task is completed, for example Hidden.
- **Early Appearance** - how the items will look if the task is started before the planned time, for example Transparent Yellow.
- **Late Appearance** - how the items will look if the task is started after the planned time, for example Transparent Red.
- **Simulation Start Appearance** - how the items should be displayed at the start of a simulation.

NOTE As a minimum, you need to define the **Simulation Start Appearance**; this assigns a unique start appearance to your task type. If you do not configure this field, the start appearance set in **Default Simulation Start Appearance** in the [Appearance Definitions Dialog Box](#) (page 632) will be used instead.

To delete a task type definition

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 Click the **Configure** tab, click the task type definition, and click the **Delete** button.

To add an appearance definition

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Click the **Configure** tab, and click **Appearance Definitions**.
- 3 Click **Add**
- 4 A new appearance definition is added to the bottom of the list; it will be highlighted, enabling you to enter a new name for it.
- 5 In the **Transparency** field, use the slider to set the transparency level, between 0 and 100 (where 0 is opaque and 100 is fully transparent).
- 6 In the **Color** field, double-click the color to open the color selector. From here, either select one of the basic colors available, or click **Define Custom Colors** to define your own color choice.
- 7 Click **OK**.

To delete an appearance definition

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Click the **Configure** tab, and click the **Appearance Definition** button.
- 3 In the **Appearance Definitions** dialog box, select the appearance definition, and click **Delete**.
- 4 Click **OK**.

To change the default simulation start appearance

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  , and click the **Configure** tab.
- 2 Click the **Appearance Definitions** button. The **Appearance Definitions** dialog box opens.
- 3 In the **Default Simulation Start Appearance** drop-down, select the appearance option, which you want to use to start your simulations.
- 4 Click **OK**.

Export

You can export a static image at any time during a simulation by using one of the standard Autodesk Navisworks image exports, whilst the **Simulation** tab is active. See [Export an Image](#) (page 494) for more information.

You also have an option to export an animation of an entire **TimeLiner** simulation. See [Export an Animation](#) (page 496) for more information.

Add Animation

In this section, you will learn how to add animation to the **TimeLiner** schedules.

Overview

You can link object and viewpoint animations to your construction schedules, and enhance the quality of simulations. For instance, you can start your simulation with a camera showing an overview of the whole project, and then, as the tasks are simulated, zoom into specific areas for a detailed view of the model. You can also play animation scenes as the tasks are simulated. For example, you can animate the arrival and depletion of the material stock piles together with the vehicle movements, and monitor the vehicular access to site.

You can add animation to an entire schedule, to individual tasks within a schedule, or combine these methods together to achieve the desired effect.

To record animation, see [Create and Edit Viewpoint Animations](#) (page 471) and [Create Object Animations](#) (page 526).

It is also possible to add scripts to the tasks in your schedule. By doing this, you gain control over the animation properties. For example, you may be able to play different animation segments as the tasks are simulated, or play animations backwards and so on.

To record scripts, see [Add Interactivity](#) (page 540).

Add Animation to an Entire Schedule

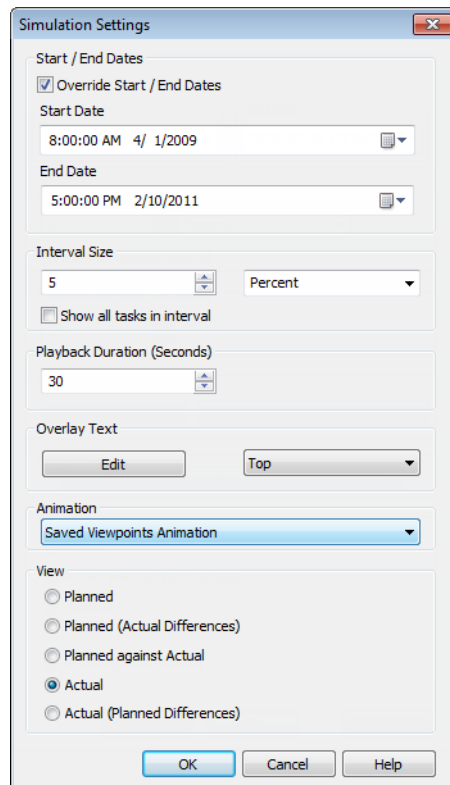
The animation that you can add to an entire schedule is restricted to viewpoints, viewpoint animations, and cameras.

Added viewpoint and camera animation will be automatically scaled to match the playback duration.

Once you've added an animation to your schedule, you can [simulate](#) (page 660) it.

To add a current viewpoint or a viewpoint animation

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner**  .
- 2 Select the required viewpoint or viewpoint animation on the **Saved Viewpoints** dockable window.
- 3 In the **TimeLiner** window, click the **Simulation** tab, and click the **Settings** button.
- 4 In the **Simulation Settings** dialog box, click the drop-down arrow in the **Animation** field, and select **Saved Viewpoints Animation**.



5 Click **OK**.

To add a camera animation

- 1 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 2 Click the **Simulation** tab, and click the **Settings** button.
- 3 In the **Simulation Settings** dialog box, click the drop-down arrow in the **Animation** field, and select the desired camera animation, for example Scene1 ► Camera.
- 4 Click **OK**.

Add Animation to Tasks

The animation that you can add to individual tasks in **TimeLiner** is restricted to scenes and animation sets from the scenes.

By default, any added animation is scaled to fit the task duration. You also got an option of playing an animation at its normal (recorded) speed, by matching its start or end point with the task.

NOTE Animation keyframes may contain transparency and color overrides. During the **TimeLiner** simulation, the transparency and color override data from **Animator** are ignored.

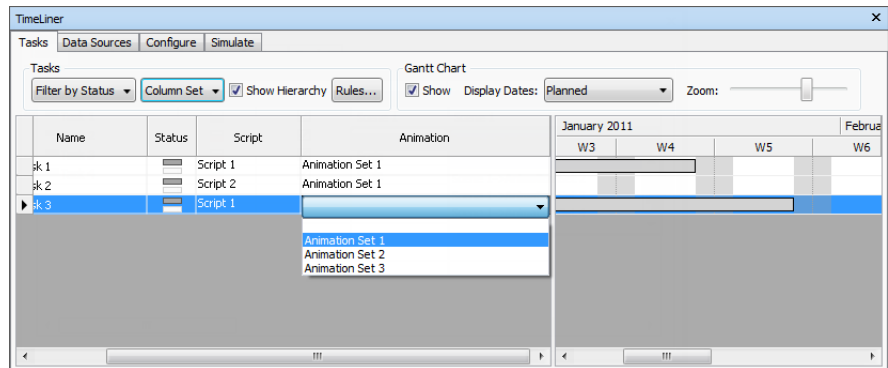
Once you've added animation to your tasks, you can [simulate](#) (page 660) your schedule.

To add an animation scene or an animation set

- 1 If the **TimeLiner** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **TimeLiner** .

- 2 On the **Tasks** tab, click the task you want to add an animation to, and use the horizontal scroll bar to locate the **Animation** column.

If you prefer, you can move the animation columns in front of the **Start Date** column as shown below.



- 3 Click the drop-down arrow in the **Animation** field, and select a scene or an animation set from a scene. When you select a scene, all animation sets recorded for this scene will be used.

- 4 Click the drop-down arrow in the **Animation Behavior** field, and select how the animation will play during this task:
 - **Scale** - the duration of the animation is matched to the duration of the task. This is the default setting.
 - **Match Start** - the animation starts when the task starts. If the animation runs past the end of the **TimeLiner** simulation, the end of the animation will be clipped.
 - **Match End** - the animation starts early enough so that it ends just when the task end. If the animation starts before the beginning of the **TimeLiner** simulation, the start of the animation will be clipped.

Add Scripts to Tasks

When you add scripts to **TimeLiner** tasks, the script events are ignored, and the script actions are run regardless of the events.

Using scripts enables you to control how the animation will play (forwards, backwards, a segment at a time and so on). You can also use scripts to change the camera viewpoints for individual tasks, or play several animations at the same time.

NOTE Before you [simulate](#) (page 660) your schedule, be sure to enable animation scripts in your file, by clicking the **Enable Scripts**  button on the **Animation** tab ➤ **Scripts** panel.

To add a script

- 1 If the **TimeLiner** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **TimeLiner** .
- 2 On the **Tasks** tab, click the task you want to add a script to, and use the horizontal scroll bar to locate the **Script** column.
- 3 Click the drop-down arrow in the **Script** field, and select the desired script to run with this task.

Locate and Manage Interferences

15

The **Clash Detective** tool enables you to search through your total project model, identifying cross-discipline interferences (clashes) earlier in the design process.

Overview of Clash Detective Tool

The **Clash Detective** tool enables effective identification, inspection, and reporting of interferences (clashes) in a 3D project model.

Using Clash Detective can help you to reduce the risk of human error during model inspections.

Clash Detective can be used as a one-off “sanity check” for a completed design work, or it can be used as an ongoing audit check of the project.

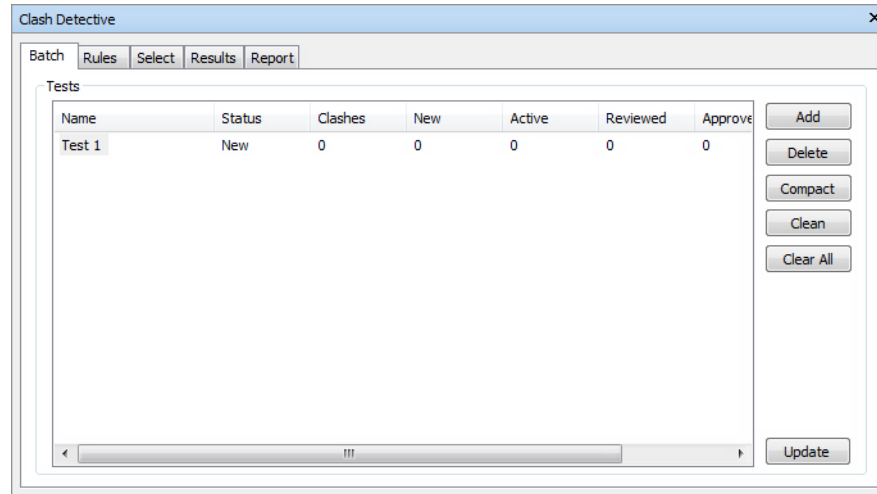
You can use **Clash Detective** to conduct clash tests between traditional 3D geometry (triangles), and laser-scanned point clouds.

You can combine the **Clash Detective** functionality with other Autodesk Navisworks tools:

- Linking **Clash Detective** and Object Animation together gives you the ability to automatically check interferences between moving objects. For example, linking a **Clash Detective** test to an existing animation scene would automatically highlight clashes for both static and moving objects during the animation, for example, a crane rotating through the top of a building, a delivery lorry colliding with a workgroup, and so on.
- Linking **Clash Detective** and **TimeLiner** together enables the time-based clash checks on the project. See [Select Items for Testing](#) (page 696) for further details on time-based clashing.
- Linking together **Clash Detective**, **TimeLiner**, and Object Animation enables clash testing of fully animated **TimeLiner** schedules.

Clash Detective Window

The **Clash Detective** dockable window enables you to set up the rules and options for your clash tests, view the results, sort them, and produce clash reports.



Clash Summary Information

The **Batch** tab displays summary information for each of the displayed clash tests. The total number of clashes in each test is displayed, with the number of clashes at each clash status identified.

The screenshot shows the 'Clash Detective' window with the 'Batch' tab selected. The window has a title bar with a close button. Below the title bar are tabs: 'Batch', 'Rules', 'Select', 'Results', and 'Report'. The 'Batch' tab contains a table with the following columns: Name, Status, Clashes, New, Active, Reviewed, Approved, and Resolved. There are four rows of data.

Name	Status	Clashes	New	Active	Reviewed	Approved	Resolved
01 Level Structure v ...	Done	0	0	0	0	0	0
02 Level Structure v ...	Done	4	4	0	0	0	0
03 Level Structure v ...	Done	81	76	3	1	1	0
04 Level Structure v ...	Done	76	43	1	0	1	31

A summary of the currently selected clash test is displayed at the top of the other **Clash Detective** tabs. The total number of clashes in the test is displayed with details of open (New, Active, Reviewed) and closed (Approved, Resolved) clashes.

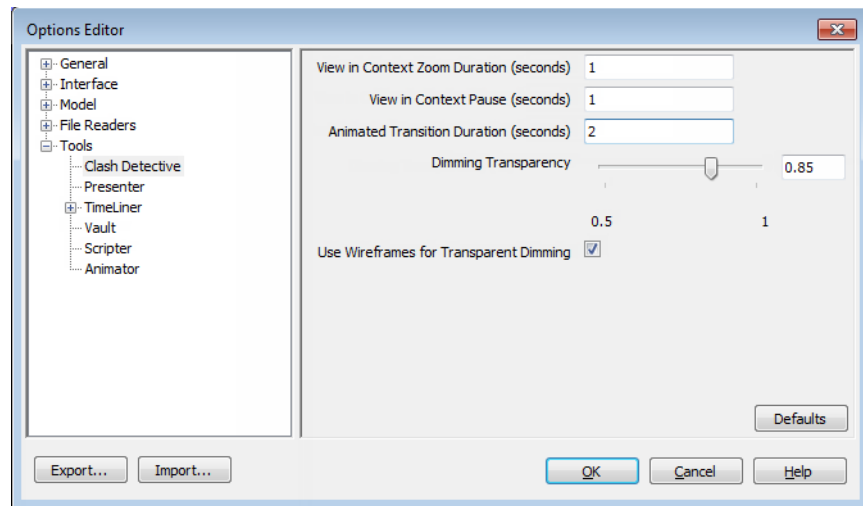
To toggle the Clash Detective window

- Click **Home** tab ➤ **Tools** panel ➤ **Clash Detective** .

 **Menu:** Classic user interface: Tools ➤ Clash Detective

To set Clash Detective options

- 1 Click the application button  ➤ **Options**.
- 2 Expand the **Tools** node in the **Options Editor**, and click the **Clash Detective** option.



- 3 On the **Clash Detective** page, use the **View in Context Zoom Duration (seconds)** box to enter the required value. When you use the **View in Context** functionality on the **Results** tab of the **Clash Detective** window, this value specifies how long it takes for the view to zoom out (using an animated transition).
- 4 Use the **View in Context Pause (seconds)** box to enter the required value. When performing a **View in Context**, the view remains zoomed out as long as you hold the button. If you quickly click rather than press and hold the button, this value specifies how long the view stays zoomed out so that the transition is not cut in mid-flow.
- 5 Use the **Animated Transition Duration (seconds)** box to enter the required value. When you click on a clash in the results grid of the **Clash**

Detective window, this value is used to smooth transitions from the current view to the next.

NOTE This option is only applicable when the **Animate Transitions** check box is selected on the **Results** tab of the **Clash Detective** window.

- 6 Use the **Dimming Transparency** slider to specify the level of transparency of items not involved in clashes.

NOTE This option is only applicable when both the **Dim Other** and **Transparent Dimming** check boxes are selected on the **Results** tab of the **Clash Detective** window.

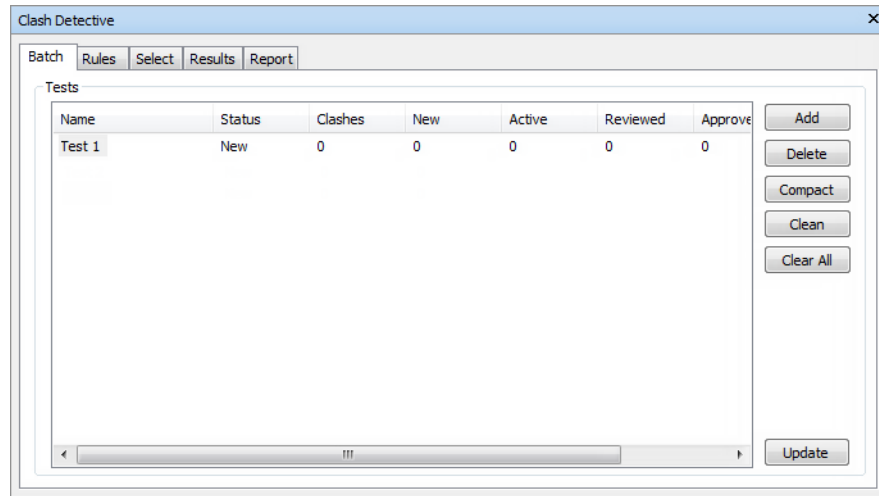
- 7 Select the **Use Wireframes for Transparent Dimming** check box to display items not involved in clashes as wireframes.

NOTE This option is only applicable when both the **Dim Other** and **Transparent Dimming** check boxes are selected on the **Results** tab of the **Clash Detective** window.

- 8 Click **OK**.

Batch Tab

The **Batch** tab enables you to manage your clash tests and results. It shows all of the clash tests currently set up, listed in a table format, and a summary of the status of all clash tests. You can use the scroll bars at the right and at the bottom of the tab to move through the clash tests.



You can also change the sort order of clash tests. To do this, click the heading of the desired column. This alternates the sort order between ascending and descending.

Buttons

Use the buttons on the **Batch** tab to set up and manage batches of clash tests.

Add - adds a new clash test to the current batch.

Delete - deletes the selected test from the current batch.

Compact - deletes all resolved clashes in the selected test.

Clean - resets the status of each clash in the selected test to 'New'.

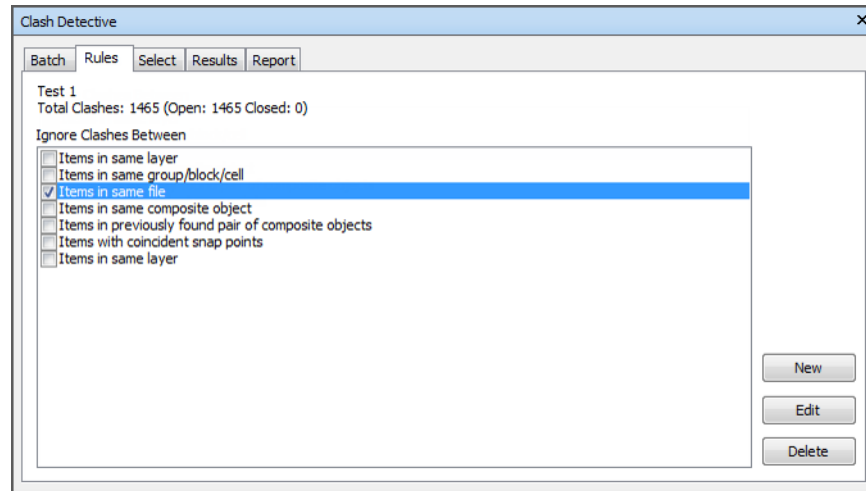
Clear All - deletes all clash tests in the batch, and displays a single empty clash test.

Update - runs all clash tests in the current batch.

Rules Tab

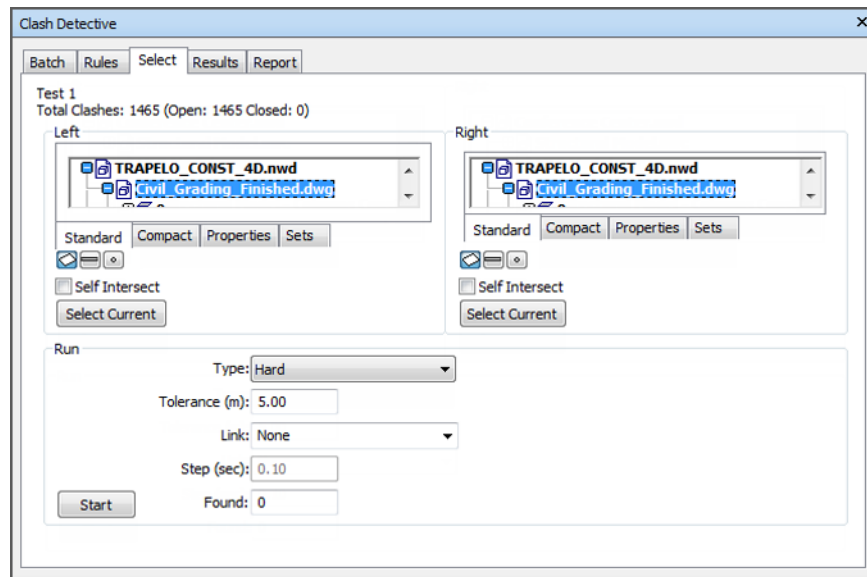
The **Rules** tab enables you to define and customize ignore rules to be applied to clash tests. It lists all currently available rules. These can be used to make **Clash Detective** ignore certain model geometry during a clash test. Each of the default rules can be edited and new rules may be added as necessary.

NOTE A summary of the currently selected clash test is displayed at the top of the tab.



Select Tab

The **Select** tab enables you to define your clash test by only testing sets of items at a time, rather than the whole model against itself. Use it to configure the parameters for the clash currently selected on the **Batch** tab.



TIP To run all tests in your batch, use the **Update** button on the **Batch** tab.

NOTE Hidden items are not included in a clash test. A summary of the currently selected clash test is displayed at the top of the tab.

The Left and Right Panes

These panes contain a Tree View of two sets of items that will be tested against each other during the clash test, and you need to select items in each pane.

At the bottom of each pane, there are several tabs replicating the current state of the **Selection Tree** window. You can use them to select items for clash tests:

- **Standard** - displays the default tree hierarchy, including all instancing.
- **Compact** - a simplified version of the tree hierarchy.
- **Properties** - the hierarchy based on the items' properties.
- **Sets** - shows the same items as on the **Sets** window.

Clash testing can be faster, more effective, and easily repeatable if you use selection and search sets. Carefully consider what sets of objects need clashing against each other and create Selection/Search Sets accordingly.

Geometry Type Buttons

Clash tests can include the clashing of surfaces, lines, and points of the selected items.

 - clashes item surfaces. This is the default option.

 - clashes items with center lines, for example pipes.

 - clashes (laser) points.

The Self Intersect Check Box

Select this check box if you want to test the geometry selection in the pane against itself, in addition to testing it against a geometry selection in the other pane.

The Select Current Button

You can also select geometry for clash testing directly in the **Scene View**, and in the **Selection Tree** dockable window.

When you have selected the required items (press and hold CTRL to select multiple objects), click the **Select Current** button under the desired pane to create the corresponding clash set.

The Shortcut Menu

Right-clicking in the **Left** or **Right** pane opens a shortcut menu:

Import Current Selection - has the same effect as pressing the **Select Current** button.

Select - selects the items in the **Scene View**.

The Run Controls

Type Selects a clash type. There are four possible types of clashes:

- **Hard** - two objects actually intersect.
- **Hard (Conservative)** - two objects are treated as intersecting even though the geometry triangles do not.

NOTE By default, this option is turned off. To switch it on, click the application



button **> Options**. In the **Options Editor**, expand the **Interface** node, click **Developer**, and select the **Show Internal Properties** check box.

- **Clearance** - two objects are treated as intersecting when they come within a specified distance of each other. Selecting this clash type also detects any hard clashes.

You might use this type of clash when pipes need to have space for insulation around them, for example.

- **Duplicates** - to intersect two objects must be identical both in type and position.

This type of clash testing may be used to clash the entire model against itself. This enables you to detect any items in the scene that may have been duplicated by mistake.

Tolerance Controls the severity of the clashes reported and the ability to filter out negligible clashes, which can be assumed to be worked around on site. Tolerance is used for Hard, Clearance and Duplicate types of clash test.

Link Enables you to link a clash test to **TimeLiner** schedule or an object animation scene.

Step Enables you to control the **Interval Size** to use when looking for clashes in a simulation sequence.

Start Runs the selected clash test.

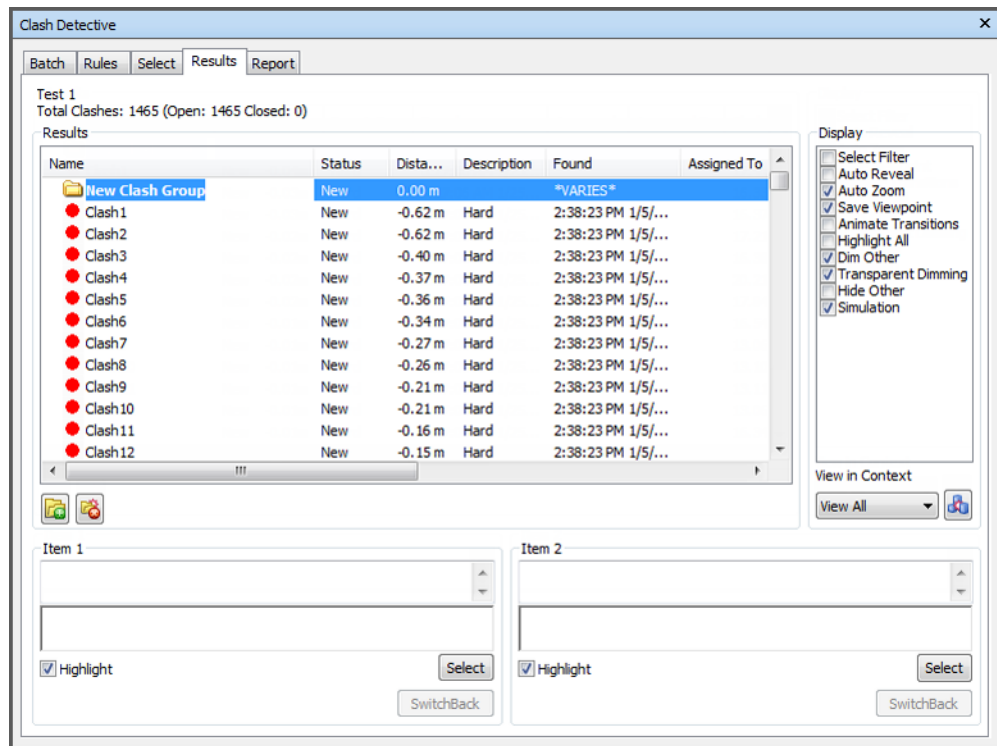
Found Displays the number of clashes found.

Results Tab

The **Results** tab enables you to interactively review the found clashes. It contains the list of clashes, and a number of controls to manage them.

You can group clashes into folders and subfolders to make managing large numbers of clashes or related clashes simpler.

NOTE A summary of the currently selected clash test is displayed at the top of the tab.



The Results Area

The found clashes are shown in a multi-column table. You can use the scroll bars at the right and at the bottom of the **Results** tab.

By default, clashes are numbered and sorted by [severity](#) (page 833).

You can sort and resize columns, if necessary.

NOTE If a clash test has been altered in some way since being set up (this might include changing an option, or having loaded the latest revision of the model) then a warning will be displayed that 'Results may not reflect the latest model or settings'. You can check tests against the latest model by re-running the test on the **Select** tab, or updating all tests on the **Batch** tab.

The Clash Icons

An icon is displayed to the left of each clash name. It visually identifies a clash status as follows:

- - **New**
- - **Active**
- - **Reviewed**
- - **Approved**
- - **Resolved**

The Clash Status

Each clash has a status associated with it. This status is updated automatically by **Clash Detective** each time the same test is run; you can also update the status yourself.

- **New** - a clash found for the first time in the current run of the test.
- **Active** - a clash found in a previous run of the test and not resolved.
- **Reviewed** - a clash previously found and marked by someone as reviewed.
This could be used in conjunction with the assign to feature, see [Assign a Clash](#) (page 707).
- **Approved** - a clash previously found and approved by someone.
If the status is manually changed to “Approved”, the user currently logged on is recorded as the approver, and the current system time is used as the time of approval.
If the test is run again, and the same clash is found, its status stays as 'Approved'.
- **Resolved** - a clash found in a previous run of the test and not in the current run of the test. The issue is, therefore, assumed to be resolved by changes to the design file and automatically updated to this status.
If the status is manually changed to “Resolved”, and a new test finds the same clash, its status will be changed back to “New”.

The Clash Group Buttons

New Clash Group  - creates a new empty clash group. By default it's called “New Clash Group (x)”, where “x” is the latest available number.

Explode Group  - ungroups the selected group of clash results.

The Shortcut Menus

Right-clicking a clash in the **Results** tab opens the following shortcut menu:

- **Add Comment** - adds a comment to the selected clash.
- **Assign** - opens the [Assign Clash Dialog Box](#) (page 741).
- **Focus on Clash** - resets the clash viewpoint so that it is focussed on the original clash point, if the original point has been navigated away from.
- **Rename** - renames the selected clash.
- **Group** - groups together all selected clashes. A new folder is added. By default it's called **New Clash Group (x)**, where x is the latest available number.

NOTE You need to select multiple items to be able to use this option.

- **Unassign** - unassigns the selected clash.

Right-clicking a clash group in the **Results** tab opens the following shortcut menu:

- **Assign** - opens the [Assign Clash Dialog Box](#) (page 741).
- **Collapse/Expand** - expands or collapses the clash group folders.
- **Add Comment** - adds a comment to the selected group.
- **Focus on Clash** - resets the clash viewpoint so that it is focussed on the original clash point, if the original point has been navigated away from.
- **Rename** - renames the selected group.
- **Explode Group** - ungroups the selected group of clash results.
- **Explode Groups** - only available when you select multiple groups. Ungroups all of the selected groups of clash results.
- **Unassign** - unassigns the selected clash group.

Right-clicking a clash, which is part of a clash group in the **Results** tab opens the following shortcut menu:

- **Assign** - opens the [Assign Clash Dialog Box](#) (page 741).
- **Add Comment** - adds a comment to the selected clash.
- **Focus on Clash** - resets the clash viewpoint so that it is focussed on the original clash point, if the original point has been navigated away from.
- **Rename** - renames the selected clash.

- **Remove from a Group** - removes the selected clashes from the clash group.
- **Unassign** - unassigns the selected clash.

The Display Area

Use the following options to enable efficient reviewing of clashes:

Select Filter Select this check box to show only the clashes that involve the items you've selected in the **Scene View** in the **Results** tab. If a clash group does not contain any clashes involving the selected item, the whole group and its contents are hidden from view.

Clear this check box to display all clash results in the **Results** tab.

NOTE Empty group folders always remain visible. If a group contains any clash involving the selected item, the group (and all clashes contained therein) remain visible. However, any individual clashes in the group that are not directly involved with selected item are displayed in italics.

Auto Reveal For an individual clash, selecting this check box temporarily hides anything obstructing the clashing items so that you can see the selected clash when zooming in on it, without having to move location.

For a clash group, selecting this check box automatically shows the 'worst' clash point in the group in the **Scene View**.

Auto Zoom Selecting this check box automatically zooms the camera to show all items involved in the selected clash or selected clash group.

Clearing this check box allows you to keep the main viewpoint static while flicking through the clashes one by one.

Save Viewpoint Selecting this check box saves the current view of the clash or a clash group so that when the clash or the clash group is reselected, the saved view is displayed.

Animate Transitions Select this check box if you require animated transitioning between clashes. When you click on a clash in the **Results** tab, the view transitions smoothly from the current view to next. You can use the **Options Editor** to customize the duration of animated transitions.

By default, this check box is clear.

TIP To benefit from this effect you need to also select the **Auto Zoom** check box or the **Save Viewpoint** check box.

Highlight All Select this check box to highlight all found clashes in the **Scene View**.

Dim Other Select this check box to turn all items that are not involved in the selected clash or selected clash group to gray. This makes it easier to see the clashed items.

Transparent Dimming This check box is only available when you select the **Dim Other** check box.

Select this check box to render all items that are not involved in the clash transparent, as well as gray. You can use the **Options Editor** to customize the level of dimming transparency, and to select to display items not involved in the clash as wireframes. By default, 85% transparency is used.

Hide Other Select this check box to hide all other items except for all those involved in the selected clash or selected clash group. This enables you to focus better on the clashing items.

NOTE Selecting this check box makes the **Dim Other** option unavailable. Clear this check box if you want to use transparent dimming.

Simulation Select this check box to use time-based and soft (animation) clashing. It moves the playback slider in the **TimeLiner** sequence or an animation scene to the exact point at which the clash occurs, and enables you to investigate the events happening immediately before and after the clash. For clash groups, the playback slider is moved to the point in time of the 'worst' clash from the group. See [Time-Based and Soft Clash Test Results](#) (page 714).

The View in Context Area

Use the options in this area to adjust the way clash results are displayed in the **Scene View**.

View in Context The options in this box enable you to temporarily zoom out to a point of reference in the model, giving context to the clash location. Choose one of the following options:

- **View All** - the view zooms out to make the whole scene visible in the **Scene View**.
- **View File Extents** - the view zooms out (using an animated transition) such that the extents of the files containing the items involved in the selected clash are visible in the **Scene View**.
- **Go to Home View** - takes you to the previously defined [Home View](#) (page 282).



Press and hold the **View in Context** button to show the chosen context view in the **Scene View**.

NOTE The view remains zoomed out for as long as you hold this button. If you quickly click (rather than press and hold) the button, the view zooms out, holds briefly, and then immediately zooms back again.

The Item 1 and Item 2 Panes

These panes contain data on both clashing items in the clash selected in the Results Area.

This includes the **Quick Properties** relating to each item in the clash and also the path through the standard **Selection Tree** from the root to the geometry of the item.

Right-clicking in the **Left** or **Right** pane opens a shortcut menu:

- **Select** - selects the items in the **Scene View**, replacing any current selection.
- **Import Current Selection** - the items currently selected in the **Scene View** become selected in the tree, if the items exist in the currently visible hierarchy.
- **Group Clashes Involving Item** - creates a new **Clash Group** containing all clashes involved with the item or items that you have right-clicked on.

The Highlight Check Box

Select this check box to override the color of the item in the **Scene View** with the color of the status of the selected clash.

The Select Button

Selecting an item in the **Item 1** or **Item 2** area, and then clicking this button selects a clashing item in the **Scene View** and in the **Selection Tree**.

The SwitchBack Button

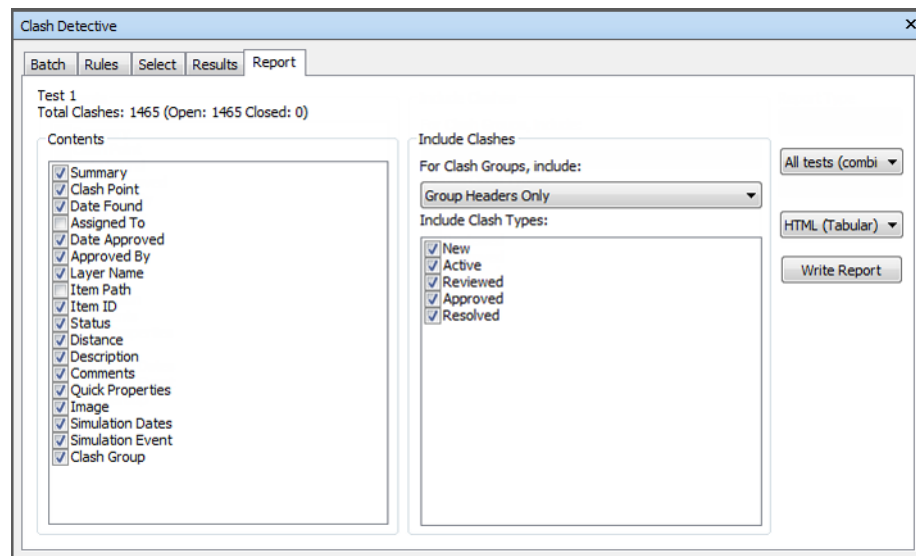
Selecting an item in the **Item 1** or **Item 2** area, and then clicking this button allows the current view and currently selected object to be sent back to the originating CAD package (AutoCAD, Revit 2012, and Microstation-based applications only).

NOTE When multiple items are selected on the tree, this button is not available.

Report Tab

The **Report** tab enables you to set up and write reports containing details of all clash results found in the selected test.

NOTE A summary of the currently selected clash test is displayed at the top of the tab.



The Contents Area

Select the required check boxes to specify which clash-related data to include in your report.

For example, you can include Quick Properties relating to the items involved in the clash, **TimeLiner** task information, image of the clash and so on.

The Include Clashes Area

For Clash Groups, Include Use the options in this box to specify how to display the clash groups in your report. Choose from the following options:

- **Group Headers Only** - reports will only include summaries of the clash group folders that you have created.

- **Individual Clashes Only** - reports will only contain individual clash results. For each clash that belongs to a group, an extra field, called **Clash Group**, can be added to the report to identify it. To enable this functionality, select the **Clash Group** check box in the **Contents** area.
- **Everything** - reports will contain both summaries of the clash group folders that you have created and individual clash results. For each clash that belongs to a group, an extra field, called **Clash Group**, can be added to the report to identify it. To enable this functionality, select the **Clash Group** check box in the **Contents** area.

NOTE If your test does not contain any clash groups, this box is not available.

Include Clash Types Select the check boxes in this field to specify which clashes to include in your report.

The Report Type

Select the type of report from the drop-down list:

- **Current Test** - creates a single report for the current test only.
- **All Tests (Combined)** - creates a single report for all tests.
- **All Tests (Separate)** - creates a separate report for each test.

The Report Format

Select the format of the report from the drop-down list:

- **XML** - creates an XML file.
- **HTML** - creates an HTML file with clashes listed sequentially.
- **HTML (Tabular)** - creates an HTML (tabular) file with the clash test(s) displayed as a table. This report can be opened and edited in Microsoft Excel 2007 onwards.
- **Text** - creates a TXT file.
- **As Viewpoints** - creates a folder called [Test_Name] on the **Saved Viewpoints** dockable window; it contains each clash saved as a viewpoint with an attached comment describing the clash.

NOTE When using XML, HTML, or Text format options consider the following:

- By default, **Clash Detective** attempts to include a JPEG viewpoint image for each clash. Make sure to select the **Image** check box in the **Contents** box, otherwise the report will contain broken image links.

- For clash groups, the viewpoint image is the aggregate viewpoint for the group.
- You need to create a separate folder for the report and its viewpoint images.

The Write Report Button

Creates the selected report and saves it into the selected location.

Set Up and Run a Clash Test

To set up and run a clash test

- 1 Select a previously run test from a [batch](#) (page 686), or start a new test.
- 2 Set the [rules](#) (page 694) for the test.
- 3 [Select](#) (page 696) the required items to be included in the test, and set the test type options.
- 4 Review the [results](#) (page 703) and assign issues to responsible parties.
- 5 Produce a [report](#) (page 717) of the identified issues and circulate for review and resolution.

Clash Batches

In this section you will learn how to set up and manage tests and test batches.

Run Clash Tests

To run clash tests

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .
- 2 Click the **Batch** tab.
- 3 To run *all* of the tests in the batch, click the **Update** button.
 To run a single test from a batch, select it in the Tests Area, click the [Select Tab](#) (page 674), and click the **Start** button.

Manage Batches of Clash Tests

To manage batches of clash tests

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .

- 2 Click the **Batch** tab.

NOTE A summary of the status of all clash tests is displayed, enabling you to quickly identify 'open' or unresolved issues.

- 3 Use the buttons to manage the tests:
 - Click **Add** to append a new test to the current batch.
 - Click **Delete** to delete the currently selected test in the **Tests Area** from the batch.
 - Click **Compact** to delete all clash results with a [clash status](#) (page 833) of **Resolved** from the test to create a smaller file.
 - Click **Clean** to reset all tests so that they are as if you had not yet run them. In other words, this will make their [clash test status](#) (page 833) **New**.
 - Click **Clear All** to remove all tests from the batch in order to start from scratch.

NOTE You can rename a test by selecting it and either pressing F2, or by clicking again the highlighted text. You can also sort the columns in the **Tests** area alphanumerically by clicking the column heads. Clicking again toggles between ascending and descending order.

Merge Clash Tests from Multiple Files

When you merge files (**Home** tab ➤ **Project** panel ➤ **Merge** ) , Autodesk Navisworks combines all of the clash test data. No duplicate geometry from the files will be loaded.

See also:

[Merge Files](#) (page 225)

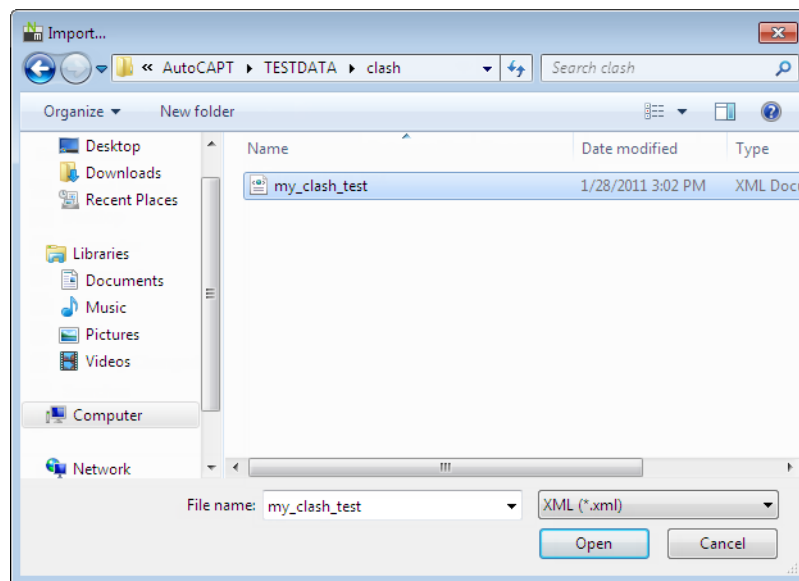
Import Clash Tests

Clash tests can be imported into Autodesk Navisworks, and used to set up pre-defined, generic clash tests.

NOTE If a clash test you want to import contains a Search Set as one of the clash selections, then the Search Set will also be imported along with all other test rules, options and selection information.

To import a clash test

- 1 Click the application button  **> Import > Clash Tests** .
- 2 In the **Import** dialog box, browse to the folder containing the clash test data, select it, and click **Open**.



Export Clash Tests

Tests can be set up to clash items based on generic properties, including direct property selection on the [Left and Right Panes](#) (page 674) of the **Select** tab, or using pre-defined Search Sets.

For example, you may have saved a Search Set that finds all pipes of a specific size, named “100mm Pipes”. Your test then clashes all 100mm pipes against the entire model. Exporting this clash test enables you to import it into another model, which will automatically set up a clash test between 100mm pipes and the entire model.

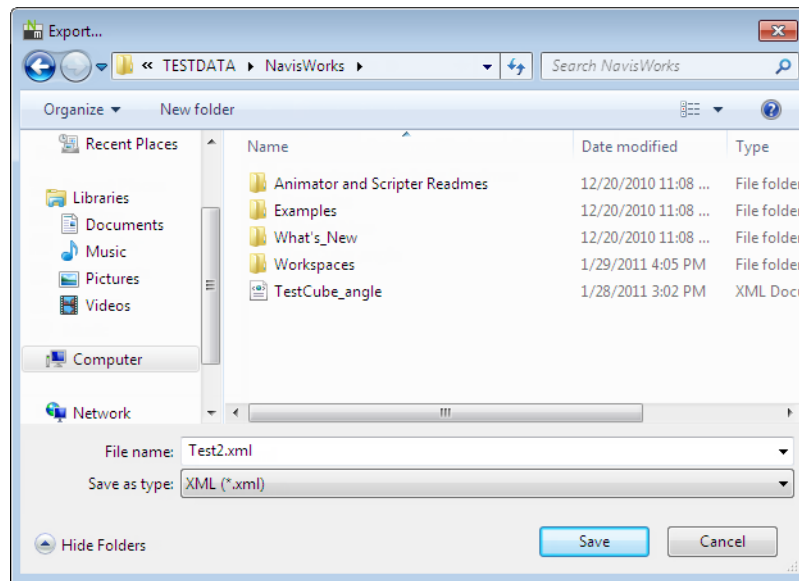
NOTE Any clash tests that are based on explicit selections will not be exported. For example, clashing one layer against another is not a valid test for exporting. To achieve this, use the **Find Items** dockable window (SHIFT+ F3) to search for each layer based on a specific property (for example: **Item, Name**). These searches can then be saved as **Search Sets**, and finally selected on the [Left and Right Panes](#) (page 674) in the **Clash Detective** window.

You can also select all loaded files on the [Left and Right Panes](#) (page 674) and **Clash Detective** will treat this as selecting the entire model.

Multiple generic tests can be set up as a batch and exported for use by other Autodesk Navisworks users, or by yourself on other projects.

To export a clash test

- 1 Click **Output** tab ► **Export Data** panel ► **Clash Tests** .
- 2 In the **Export** dialog box, enter a new file name and location, if you wish to change from those suggested.



3 Click **Save**.

Create Custom Clash Tests

Exported clash tests can be used as a basis to define custom clash tests. If you have a common set of clash tests that you reuse on multiple projects, you can turn them into a custom clash test. Once installed as a custom clash test, the entire batch of tests can be selected and run directly from the [Select Tab](#) (page 674). The results from all tests in the batch are combined and presented as the results of the custom clash test. The name of each test in the batch is displayed in the **Description** field of the results.

Custom clash tests are an excellent way to roll out a standardized set of tests across an organization. They allow the expertise of “power” users to be reused by everyone. Finally, they can be seen as a way of implementing object intelligence. For example, a custom clash test could be written that checked for compliance with a local building code based on object information and properties defined in a particular CAD system.

To define and use a custom clash test

- 1 [Export Clash Tests](#) (page 689) to an XML file. The name of the file is used as the default name for the custom test.

- 2 If you want, change the name of the custom test by editing the XML file directly. The top level element in the XML file is called “batchtest”. The name of the custom test as displayed to the user is defined by the “name” attribute. The name of the custom test as saved in a file is defined by the “internal name” attribute.
- 3 To install the custom test, copy the exported XML file to the custom_clash_tests folder of one of the Autodesk Navisworks search directories, for example: C:\Documents and Settings\All Users\Application Data\Autodesk Navisworks Manage 2012\custom_clash_tests. See [Search Directories](#) (page 159) for more information.
- 4 Restart Autodesk Navisworks. On start up, **Clash Detective** checks these search directories for custom clash tests.
- 5 To use the custom test, open the **Clash Detective** window, and click the **Select** tab.
- 6 Select your custom test from the [Type](#) (page 674) drop-down box.
- 7 Click the **Start** button. All other options and rules are specified by the custom test.

Clash Rules

Using ignore clash rules reduce the number of clash results by ignoring certain combinations of clashing items. The **Clash Detective** tool includes both the default clash rules and clash rule templates that can be used to create custom clash rules.

Default Clash Rules

The following ignore clash rules are built-in:

- **Items in Same Layer** - any items found clashing that are in the same layer are not reported in the results.
- **Items in Same Group/Block/Cell** - any items found clashing that are in the same group (or inserted block) are not reported in the results.
- **Items in Same File** - any items found clashing that are in the same file (either externally referenced or appended) are not reported in the results.
- **Items in Same Composite Object** - any items found clashing that are part of the same composite object (an item composed of multiple parts of geometry) are not reported in the results.

- **Items in Previously Found Pair of Composite Objects** - any items found clashing that are part of composite objects (items composed of multiple parts of geometry) that have previously been reported in the test are not reported in the results.
- **Items with Coincident Snap Points** - any items found clashing that have snap points that coincide are not reported in the results. This can be particularly useful for pipe runs made from cylinders.

Rule Templates

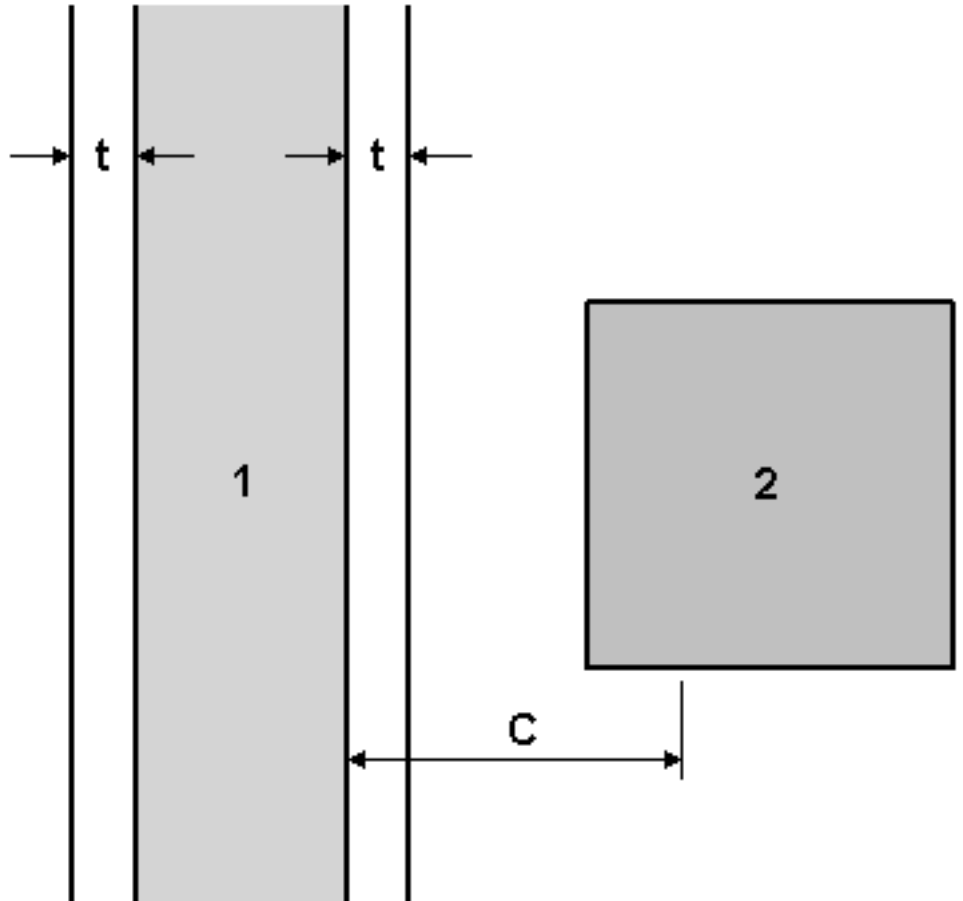
You can also create your own ignore rules. The following rule templates are available (in addition to the default clash rules):

- **Insulation Thickness** - any items found clashing that have the clearance value greater than the specified insulation thickness are not reported in the results. This rule should be used with a [Clearance](#) (page 674) test.

If you have a pipe requiring a specific thickness of insulation, then you may want to carry out a clearance test on that pipe, setting the clearance tolerance to the required insulation thickness. This would identify any areas where there is not enough clearance around the pipe to install the insulation.

If you have various pipes, all requiring different thicknesses of insulation then rather than setting up a separate clearance test for each thickness, you can set up one test with the greatest necessary tolerance, that is, assume all pipes require the maximum thickness of insulation. This rule can then be applied to ignore any clashes that are falsely identified, as their actual insulation thickness is less than the maximum clearance used.

See the diagram below for an example of where this ignore rule would be applied:



Pipe **1** has insulation thickness **t** and **C** is the maximum clearance (thickest insulation) required anywhere in the current model. Any items (**2**) that come within the range of **t** to **C** will not be reported in the results.

- **Same Property Value** - any items found clashing that share a specific property value are not reported in the results. This template can be used when the information is stored on the same property.
- **Same Selection Sets** - any items found clashing that are contained within the same Selection Set are not reported in the results.

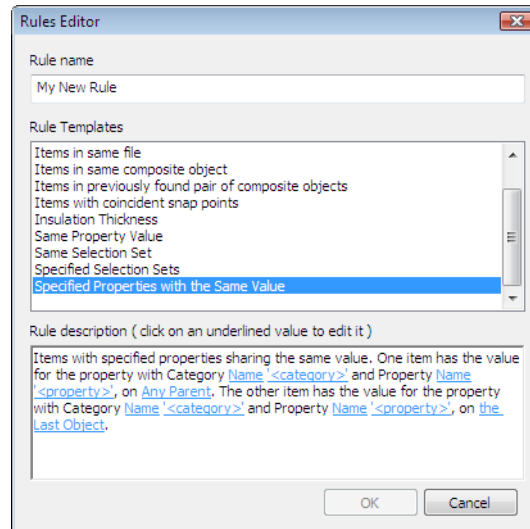
- **Specified Selection Sets** - any items found clashing that are contained within two specified Selection Sets are not reported in the results.
- **Specified Properties with the Same Value** - any items found clashing that share the same value, but on two different properties are not reported in the results. This is a new rule template. This rule also enables you to look for the property on any parent object. For example, the gasket at the end of a pipe is registered as a clash with the nozzle of pump. These items should be connected. The gasket itself does not have any property attached directly to it to identify that it should be attached to the nozzle of the pump; however the parent object of the gasket does. If you use this template, the clash test ignores clashes between these two items.

To use clash rules

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective**  .
- 2 Click the **Batch** tab, and select the test you want to configure in the Tests Area.
- 3 Click the **Rules** tab, and select the check boxes for all exception rules that you want to apply to your test.

To add a custom clash rule

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective**  .
- 2 On the **Rules** tab, click the **New** button.
- 3 In the **Rules Editor** dialog box, enter a new name for your rule.



- 4 In the **Rule Templates** list, click the template you want to use.
- 5 In the **Rule Description** box, click each of the underlined values to define your custom rule. The customizable values available with the built-in templates are:
 - **Name.** Use the name of the category or property as it is displayed in the interface (recommended). You can also choose **Internal Name** which is that accessed via the API (for advanced use only).
 - **"<category>".** Choose from the available list, which category the property you wish to define is in. Only the categories that are contained in the scene are available in the drop-down list.
 - **"<property>".** Choose from the available list, which property you want to define. Again, only the properties in the scene within the chosen category will be available.
 - **Any Parent.** Search for the defined property on the specified selection. **Any Parent** is the default option, though you can also choose **a Model, a Layer, the Last Object, or the Geometry**
 - **the Last Object.** Search for the defined property on the specified selection. **The Last Object** is the default option, though you can also choose **Any Parent, a Model, a Layer, or the Geometry**.
 - **"<set>".** Choose from the available list, which Set you require to define the rule. Only the pre-defined Selection and Search Sets are available in the drop-down list.

- 6 Click **OK**. The rule is added to the **Ignore Clashes Between** area on the **Rules** tab.

To edit a clash rule

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .
- 2 On the **Rules** tab, click the ignore rule you want to edit.
- 3 Click the **Edit** button.
- 4 In the **Rules Editor** dialog box, rename the rule if you want to change it from its current name.
- 5 Choose another **Rule Template** if you want to change it from the current template.
- 6 In the **Rule Description** box, click each of the underlined values to redefine your custom rule.
- 7 Click **OK** to save the changes you've made to the rule.

To delete a clash rule

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .
- 2 On the **Rules** tab, click the ignore rule you want to delete.
- 3 Click the **Delete** button to delete the clash rule.

Select Items for Testing

In this section you will learn how to set up various parameters for your tests.

Select Items for a Clash Test

To select items

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .
- 2 Click the **Batch** tab, and select the test you want to configure.

3 Click the **Select** tab.

There are two identical panes in this tab called **Left** and **Right**. These panes contain a Tree View of two sets of items that will be tested against each other during the clash test, and you need to select items in each. You can select the items by choosing a tab from the selection tree and manually selecting items from the tree hierarchies. Any selection sets in the scene are also included on a tab, which is a quick and useful method of setting up items across sessions (see [Interactive Geometry Selection](#) (page 337)).

You can also transfer the current selection to one of the boxes by selecting items in the usual way in the **Scene View** or **Selection Tree** and clicking the appropriate **Select Current** button.

4 Optional: Select the appropriate **Self Intersect** check box to test the corresponding set for self-intersection, as well as intersection against the other set.

5 Optional: You can include the clashing of points, or lines, or surfaces in your test. Underneath each window there are three buttons that correspond to surfaces, lines and points. To toggle a button, click it.

So, for example, if you want to run a clash test between some surface geometry and a point cloud, you can set up the geometry in the **Left** pane, and click the **Point Cloud**  button under the **Right** pane. The **Surface**  button under the **Left** pane is toggled on by default. Additionally, you could set the clash **Type** to **Clearance** with a **Tolerance** of 1 meter.

NOTE If the **Type** is set to **Hard**, lines and surfaces will actually need to intersect with any points to register a clash.

Select Clash Test Options

There are four default clash test types for you to choose from:

- **Hard**. Choose this option if you want the clash test to detect actual intersections between geometry.
- **Hard (Conservative)**. This option performs the same clash test as **Hard**, however, it additionally applies a [conservative intersection method](#) (page 838).

NOTE This clash test type is only available when in **Developer** profile. See [Profiles](#) (page 158) for more information.

- **Clearance.** Choose this option if you want the clash test to check for geometry within a specific distance from other geometry (see [tolerance](#) (page 839) for more information). You can use this type of clash when, for example, pipes need to have space for insulation around them.

NOTE Clearance clashes are not the same as “soft” clashes. Clearance clashes detect for static geometry coming within a distance of other geometry, whereas soft clashes detect potential clashes between moving components. **Clash Detective** supports soft clash checking when you link it to Object Animation.

- **Duplicates.** Choose this option if you want the clash test to detect for duplicate geometry. You can use this type of clash test to check a model against itself to ensure the same part has not been drawn, or referenced twice, for example.

To select the clash test options

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective** .
- 2 On the **Select** tab, select the test you want to run from the **Type** drop-down list. Any custom clash tests that have been defined (see [Create Custom Clash Tests](#) (page 690)) appear at the end of the list.
- 3 Enter the **Tolerance** required, which will be in [Display Units](#) (page 157). See [Clash Detective Terminology](#) (page 833) for more information.
- 4 If you want to run a time-based or soft clash test, select the appropriate option in the **Link** box. For example, selecting **TimeLiner** will make **Clash Detective** produce a clash report based on the **Clash Detective** settings, **TimeLiner** simulation settings, and the project data contained in **TimeLiner**. For more information and usage examples, see [Time-Based and Soft Clash Testing](#) (page 698).

Time-Based and Soft Clash Testing

Linking to a **TimeLiner** schedule integrates the features of **Clash Detective** and **TimeLiner** together, allowing the automation of interference checking throughout the lifecycle of a **TimeLiner** project.

Similarly, linking to an object animation scene integrates the features of **Clash Detective** and Object Animation giving you the ability to automatically check interferences between moving objects.

Finally, you can link to an animated **TimeLiner** schedule (a schedule in which some tasks are linked to animation scenes), and run automatic time-based soft clash tests.

Time-Based Clashing

Project models can include the static representation of temporary items, such as work packages, ships, cranes, installations, and so on. Such static objects can be added into the **TimeLiner** project, and scheduled to appear and disappear at particular locations, over specific period of time.

As these static package objects 'move' around a project site, based on a **TimeLiner** schedule, it is possible that some static package objects could, at some point in the schedule, take up the same space, that is 'clash'.

Setting up time-based clashing enables the automated checking of this throughout the project lifetime. When a time-based clash session is run, at each step of **TimeLiner** sequence, **Clash Detective** is used to check if a clash has occurred. If it has, a date of when the clash took place is logged, along with the event that caused the clash.

You can review the [Clash Results](#) (page 703) and, for example, reschedule objects' appearance to eliminate such clashes.

To prepare for time-based clashing

- 1 Each static package that you want to use needs to be modelled in the project model covering the area or volume required; for example, you can use semi-transparent blocks.
- 2 These static packages must be added as tasks to the **TimeLiner** schedule.
- 3 Extra task types must be added to **TimeLiner** (on the **Configure** tab) to represent different types of static packages. You also need to configure the appearance for each added task type.

To link to a TimeLiner schedule

- 1 In Autodesk Navisworks open the project model file, which contains the **TimeLiner** schedule with the static package tasks.
- 2 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 3 Click the **Tasks** tab, and check the static package tasks are displayed.

- 4 Click the **Configure** tab, and check the task types have been added to match the static packages.
- 5 Click the **Simulation** tab, and play the simulation to view the static packages being displayed. Check they are shown in the correct location and over the correct periods of time.
- 6 If the **Clash Detective** window is not already open, click **Home** tab
 ➤ **Tools** panel ➤ **Clash Detective**  .
- 7 Click the **Select** tab.
- 8 In the **Left** and **Right** panes, select the objects you want to test.
- 9 In the **Link** drop-down box, select **TimeLiner**.
- 10 Click the **Start** button. **Clash Detective** will check at each interval in the project to see if there are any clashes present. The number of found clashes is displayed in the **Found** box.

NOTE If the **TimeLiner** project is large in size, running this type of clash test will invariably take more time to complete than a normal clash run.

You can now [review](#) (page 703) the found clashes.

Soft Clashing

Project models can include the dynamic representation of temporary items, such as work packages, ships, cranes, installations, and so on.

You can use the **Animator** window to create animation scenes with these objects, so that they will be moving around a project site, or change their size, and so on. It is possible that some moving objects could collide.

Setting up soft clashing enables the automated checking of this. When a soft clash session is run, at each step of a scene sequence, **Clash Detective** is used to check if a clash has occurred. If it has, a time of when the clash took place is logged, along with the event that caused the clash.

You can review the [Clash Results](#) (page 703) and, for example, reschedule objects' movement to eliminate such clashes.

To prepare for soft-clashing

- 1 Each object that you want to animate needs to be modelled in the project model covering the area or volume required; for example, you can use semi-transparent blocks.

- 2 Animated scenes with the desired objects must be created by using the **Animator** window in Autodesk Navisworks.

To link to Object Animation

- 1 In Autodesk Navisworks open the project model file, which contains the object animation scenes.
- 2 If the **Animator** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **Animator** .
- 3 Play the animation. Check that the animated objects are shown in the correct location, in the correct size, and so on.
- 4 If the **Clash Detective** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **Clash Detective** .
- 5 Click the **Select** tab.
- 6 In the **Left** and **Right** panes, select the objects you want to test.
- 7 In the **Link** drop-down box, select the animation scene you want to link to, for example **Scene1**.
- 8 In the **Step** box, enter the **Interval Size** to use when looking for clashes in an animation.
- 9 Click the **Start** button. **Clash Detective** will check at each interval in the animation to see if there are any clashes present. The number of found clashes is displayed in the **Found** box.

NOTE If the animation scene is large in size, running this type of clash test will invariably take more time to complete than a normal clash run.

You can now [review](#) (page 703) the found clashes.

Time-Based Soft Clashing

Project models can include the representation of temporary items, such as work packages, ships, cranes, installations, and so on. If you want to use static objects, they must be added into the **TimeLiner** project, and scheduled to appear and disappear at particular locations, over specific period of time.

Additionally, you can create dynamic animation scenes, so that the objects will be moving around a project site, or change their size, and so on. Once

created, such scenes must be linked to tasks in the **TimeLiner** project schedule.

It is possible, that appearance or disappearance of static objects can obstruct the movement of animated objects on site.

Setting up a time-based soft clashing enables the automated checking of this throughout the project lifetime. When a time-based soft clash session is run, at each step of **TimeLiner** sequence, **Clash Detective** is used to check if a clash has occurred. If it has, a date of when the clash took place is logged, along with the event that caused the clash.

You can review the [Clash Results](#) (page 703) and, for example, re-route some objects to avoid any space/time conflicts.

To prepare for time-based soft clashing

- 1 You need to set up necessary [static packages](#) (page 699).
- 2 You need to set up necessary [dynamic packages](#) (page 700).
- 3 You need to link the dynamic packages (animation scenes) to the correct tasks in **TimeLiner**.

To link to a TimeLiner schedule

- 1 In Autodesk Navisworks open the project model file, which contains the animated **TimeLiner** schedule.
- 2 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** .
- 3 Click the **Tasks** tab, and check the static package tasks are displayed, and at least one animation scene has been linked to one of the **TimeLiner** tasks.
- 4 Click the **Configure** tab, and check the task types have been added to match the static packages.
- 5 Click the **Simulation** tab, and play the simulation. Check the static and dynamic packages are shown in the correct location and over the correct periods of time.
- 6 If the **Clash Detective** window is not already open, click **Home** tab ► **Tools** panel ► **Clash Detective** .
- 7 Click the **Select** tab.
- 8 In the **Left** and **Right** panes, select the objects you want to test.

- 9 In the **Link** drop-down box, select **TimeLiner**.
- 10 In the **Step** box, enter the **Interval Size** to use when looking for clashes in animation scenes.
- 11 Click the **Start** button. **Clash Detective** will check at each interval in the project to see if there are any clashes present. The number of found clashes is displayed in the **Found** box.

NOTE If the **TimeLiner** project is large in size, running this type of clash test will invariably take more time to complete than a normal clash run.

You can now [review](#) (page 703) the found clashes.

Run an Individual Clash Test

To run an individual clash test

- 1 If the **Clash Detective** window is not already open, click **Home** tab
 > **Tools** panel > **Clash Detective** .
- 2 Click the **Batch** tab, and select the test you want to run.
- 3 Click the **Select** tab, and set the required test options.
- 4 Once the left and right sets are selected and the clash type and tolerance defined, click the **Start** button to start the test running.

The **Found** box shows how many clashes have been found so far during this test.

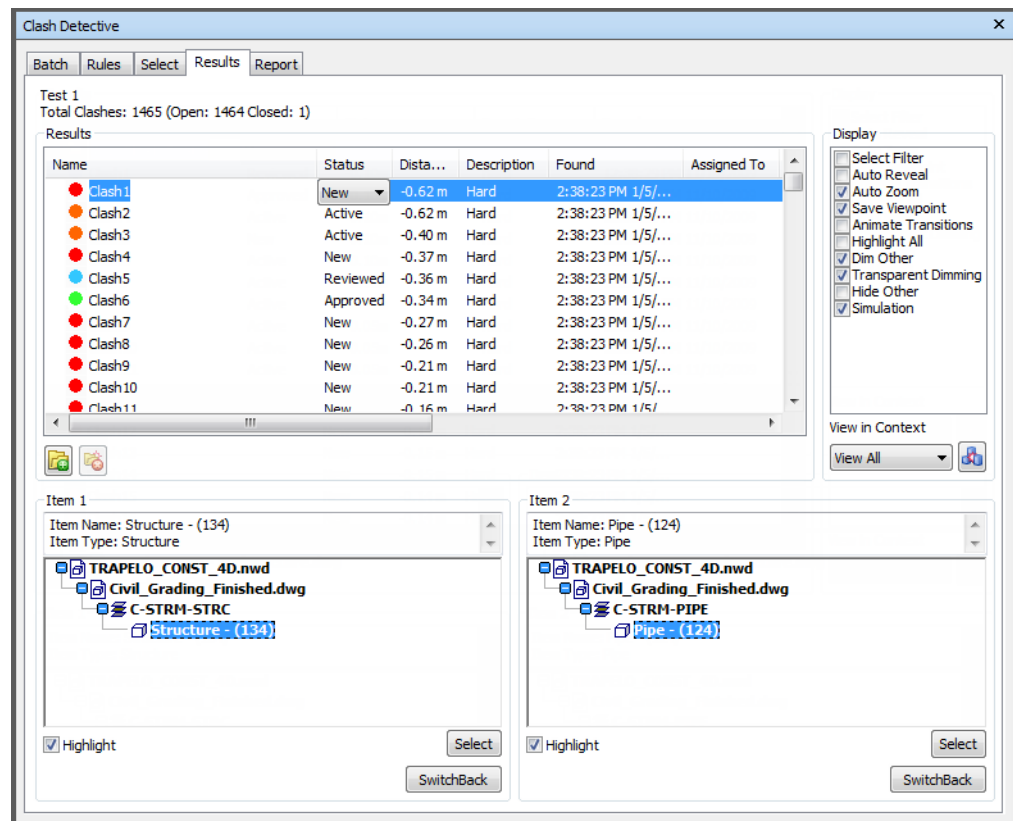
NOTE The progress bar shows how far through the test **Clash Detective** has got. If you wish to stop the test at any time, press the **Cancel** button. All clashes found up until the interrupt will be reported, and the test will be saved with a **Partial** status.

Clash Results

In this section you will learn how to interact with the results of your clash tests.

Understand Clash Results

All found clashes are shown in the **Results** tab in a multi-column table. You can click on any of the column headings to sort the table using that column's data. This sorting can be alphabetical, numerical, date-related, or, for the **Status** column, in workflow order: **New**, **Active**, **Reviewed**, **Approved**, and **Resolved**. Clicking the column heading repeatedly toggles the sorting order between ascending and descending.



If you run a time-based, soft, or time-based soft clash test, the start time/date of any clashes are recorded next to the appropriate clash under the **Start** and **End** columns, together with the event name (an animation scene or a **TimeLiner** task) in the **Event** column. If multiple clashes are found on a particular date, each individual clash is listed with identical simulation information.

The **Item 1** and **Item 2** panes show the **Quick Properties** relating to each item in the clash, and also the path through the standard **Selection Tree** from the root to the geometry of the item.

Clicking on a clash highlights the two objects involved in the clash in the **Scene View**. By default, the center of the clash is the center of the view, which is zoomed in so that the parts of objects involved in clash fill the view. You can use the options in the **Display** and the **View in Context** areas to control how the clash results are shown.

When your clash results contain multiple clashes which are associated with a single design issue, consider manually grouping them together. Organizing clashes into folders can simplify keeping track of design issues. When clashes are grouped together they are listed as a single clash in summaries and reports. The distance shown for a clash group is that of the largest 'open' clash with the group.

NOTE When you select a clash group containing both 'open' (**New, Active, Reviewed**) and 'closed' (**Approved, Resolved**) clashes, the open clashes will be highlighted in the **Item 1** and **Item 2** panes rather than the closed clashes, regardless of the actual size of the clashes.

Finally, selecting an item on the **Item 1** or **Item 2** pane, and then clicking on the **SwitchBack** button sends the current view and currently selected object back to the originating CAD package. This makes it very simple to show clashes in Autodesk Navisworks, take them back to the CAD package, alter the design, and then reload them in Autodesk Navisworks for a very fast design review turnaround.

Manage Clash Results

You can manage clash results individually. You can also create and manage clash groups. This can be useful where you have multiple clashes involving a particular item, or where a certain area contains multiple clashes that can be considered as a single issue. Created groups are represented in the **Results** tab as folders.

You can also assign clashes and clash groups to an individual or trade, so that you can specify who is responsible for resolving a clash.

To rename an ungrouped clash

- 1 On the **Results** tab, right-click the clash in the **Results** tab, and click **Rename**.

- 2 Type the new name, and press Enter.

 **Command entry:** F2

To create a clash group

- 1 Click the **New Clash Group**  button on the **Results** tab. A new folder called **Clash Group X** is added above the currently selected clash (or at the top of the list if nothing is selected).
- 2 Type in a new name for the group, and press Enter.
- 3 Select clashes you want to add to this group, and drag them into the folder.
- 4 When you click the created clash group, the **Item 1** and **Item 2** panes show all the clashing items contained within that clash group, and all corresponding clashes are shown in the **Scene View**.

To group multiple clashes

- 1 On the **Results** tab, select all clashes you want to group together.
- 2 Right-click your selection, and click **Group**.
- 3 Type in a new name for the group, and press Enter.
- 4 When you click the created clash group, the **Item 1** and **Item 2** panes show all the clashing items contained within that clash group, and all corresponding clashes are shown in the **Scene View**.

To rename a clash group

- 1 Right-click the clash group in the **Results** tab, and click **Rename**.
- 2 Type the new name, and press Enter.

 **Command entry:** F2

To remove clashes from a group

- 1 On the **Results** tab, expand the required clash group.
- 2 Select all clashes you want to remove.
- 3 Right-click the selection, and click **Remove from Group**.

To ungroup clash results

- 1 On the **Results** tab, click the desired clash group.

- 2 Click the **Explode Group**  button.

Shortcut menu: Explode Group

To change status of ungrouped clashes

- 1 On the **Results** tab, select the clashes requiring the change.
- 2 Click the drop-down button in the **Status** box, and select the appropriate status.

NOTE Multiple clashes can be changed by either holding down the CTRL key whilst selecting individual clashes (the last selected being shown in the **Scene View**), or by holding down the SHIFT key and selecting another clash, thereby selecting all clashes between the two including the first and last (again, the last selected being shown in the **Scene View**).

To change status of a clash group

- 1 On the **Results** tab, click the required clash group.
- 2 Click the drop-down button in the **Status** box, and select the appropriate status.

NOTE Changing status of a clash group changes status of all clashes in this group.

To assign a clash

- 1 On the **Results** tab, select a clash, clash group or multiple clashes.
- 2 Right-click the selection, and click **Assign**.
- 3 Type in the name of the person/trade the selection is to be assigned to.

NOTE If the clash is already assigned then the assignee will be displayed when the dialog box is opened.

- 4 Type any notes if required.
- 5 Click **OK**.

To unassign a clash

- 1 On the **Results** tab, select a clash, clash group or multiple clashes.
- 2 Right-click the selection, and click **Unassign**.

NOTE A comment is automatically written against a clash or clash group when it is unassigned.

Review Clash Results

Autodesk Navisworks Manage 2012 provides tools to add comments and redlines to clash results. If multiple clashes are associated with a single design issue, consider manually grouping them together at the beginning of your review process. See [Manage Clash Results](#) (page 705).

Visually Identify Clashes in a Model

As you click on a clash result, you are automatically zoomed into the clash location in the **Scene View**. The **Clash Detective** tool contains a number of **Display** options that enable you to adjust the way clashes are rendered in the model.

For example, you can visually isolate the clashing items in the **Scene View** making it easier to focus on them.

You can also adjust the viewing context to visually identify where each clash is within the model, and customize the way Autodesk Navisworks transitions between clashes.

When you navigate your clash results in the **Scene View**, any changes that you apply to clash viewpoints will be automatically saved, if the **Save Viewpoint** check box is selected on the **Results** tab. This option is designed to allow you to tailor the viewpoint for a clash result, and to store redlining. If you navigated away from the clash and cannot locate it in the **Scene View**, you can reset the viewpoint to focus on the clash point again.

To show only the clash results that involve your current selection

- 1 Select the desired geometry on the **Selection Tree** or in the **Scene View**.
- 2 In the **Clash Detective** window click the **Results** tab, select the **Select Filter** check box.

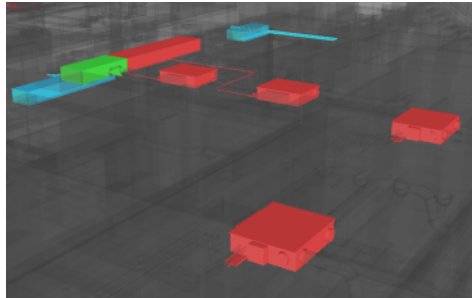
Only those clashes that involve the selected items are now shown in the **Results** area.

To change how the clashing items are highlighted

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 Select the **Highlight** check boxes for **Item 1** and **Item 2** to override the color of the clashing items in the **Scene View** with the color of the status of the selected clash.

To highlight all clashes in the Scene View

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 Select the **Highlight All** check box in the **Display** area. All found clashes are highlighted in the color of their status.

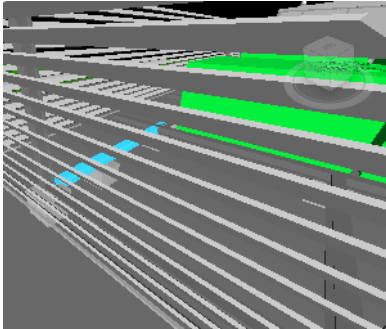


Clear the check box to return to default view of highlighting only the items involved in the clash selected in the **Results** area.

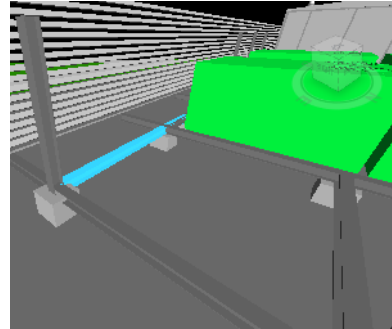
To isolate clash results in the Scene View

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 To hide all items that obstruct viewing the clashing items in the **Scene View**, select the **Auto Reveal** check box. When you click on a clash

result, you can see the clash when zooming in on it, without having to move location.

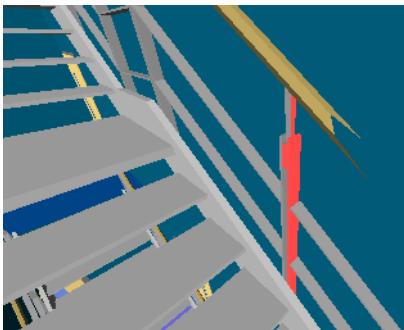


Viewing clashing items with the Auto Reveal check box clear



Viewing clashing items with the Auto Reveal check box selected

- 3 To hide all items not involved in the clash, select the **Hide Other** check box. This enables you to focus better on the clashing items in the **Scene View**.

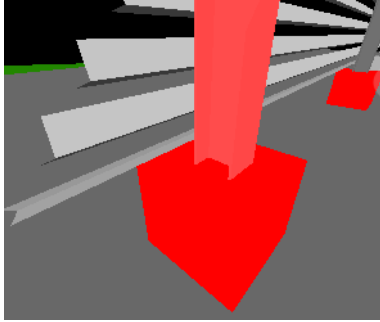


Viewing clashing items with the Hide Other check box clear

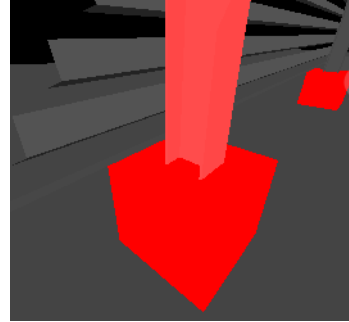


Viewing clashing items with the Hide Other check box selected

- 4 To dim all items not involved in the clash, select the **Dim Other** check box. When you click on a clash result, Autodesk Navisworks turns all items not involved in the clash to gray.



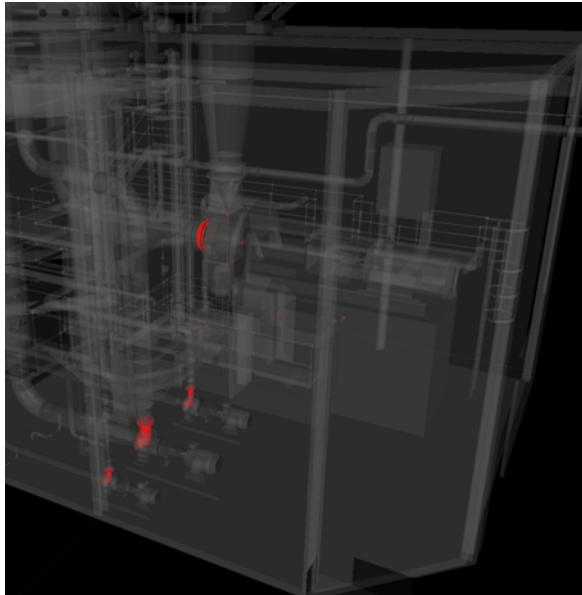
Viewing clashing items with the Dim Other check box clear



Viewing clashing items with the Dim Other check box selected

- 5 To set transparent dimming of all objects not involved in the clash, select the **Transparent Dimming** check box. This option can only be used with the **Dim Other** option, and will render all items that are not involved in the clash transparent, as well as gray. The level of dimming transparency can be customized in the **Options Editor**. By default, 85% transparency is used.

TIP Try using this option as you highlight all clashes in the **Scene View** (by selecting the **Highlight All** check box). You can see the clashes through surrounding objects.



To set up transitions between clashes

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 In the **Display** area, make sure the **Auto Zoom** check box is selected.
- 3 Click a clash result in the **Results** list. You are zoomed in to the location of a clash in the **Scene View**.
- 4 Select the **Animate Transitions** check box.
- 5 Click another clash result. The view transitions smoothly from the current view to next. You can use the **Options Editor** to customize the duration of animated transitions.

To view clashes in context

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 Click a clash result in the **Results** list.
- 3 In the **Display** area, make sure the **Auto Zoom** and **Animate Transitions** check boxes are selected.
- 4 To make the whole scene visible in the **Scene View**, select **View All** in the **View in Context** box.

To limit the view to the extents of the files containing the items involved in the selected clash, select **View File Extents** in the **View in Context** box.

- 5 Press and hold the **View in Context**  button to show the chosen context view in the **Scene View**. The view remains zoomed out for as long as you hold this button. If you quickly click (rather than press and hold) the button, the view zooms out, holds briefly, and then immediately zooms back again.

To save viewpoints with clash results

- 1 In the **Clash Detective** window, click the **Results** tab.
- 2 In the **Display** area, make sure the **Save Viewpoint** check box is selected.
- 3 Click a clash result in the **Results** list. This allows you to tailor the viewpoint for a clash result. It also enables redlining to be stored with a clash result.

NOTE Once redlining has been added, subsequent changes to the viewpoint due to navigation will not be saved. In order to save a different viewpoint, the redlining must first be removed using the redline **Erase** tool.

To reset the viewpoint to focus on the clash

- 1 Click the **Results** tab in the **Clash Detective** window.
- 2 Right-click the desired clash or clash group, and click **Focus on Clash** on the shortcut menu.

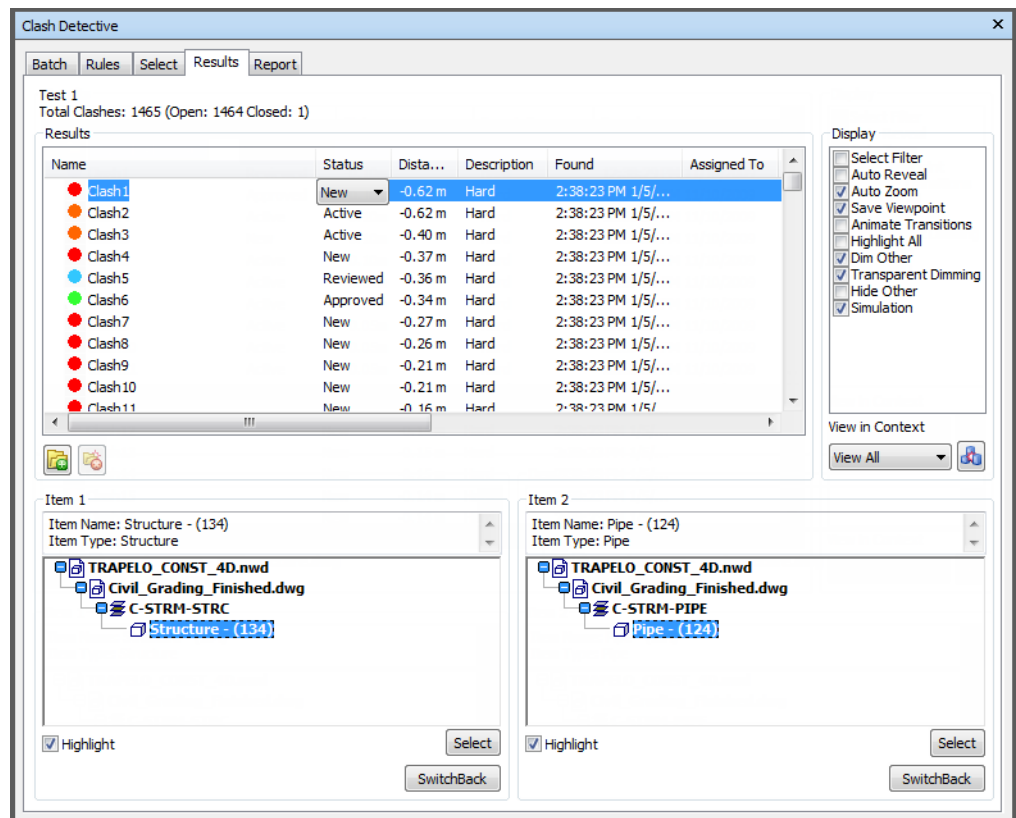
Add Review Comments and Redlines

You can add comments and redlines to clash results. For more information, see [Use Comments, Redlines, and Tags](#) (page 399).

Time-Based and Soft Clash Test Results

To review time-based clash results

- 1 In the **Clash Detective** window, set up and run a time-based clash test (see [Time-Based Clashing](#) (page 699)).
- 2 Click the **Results** tab.

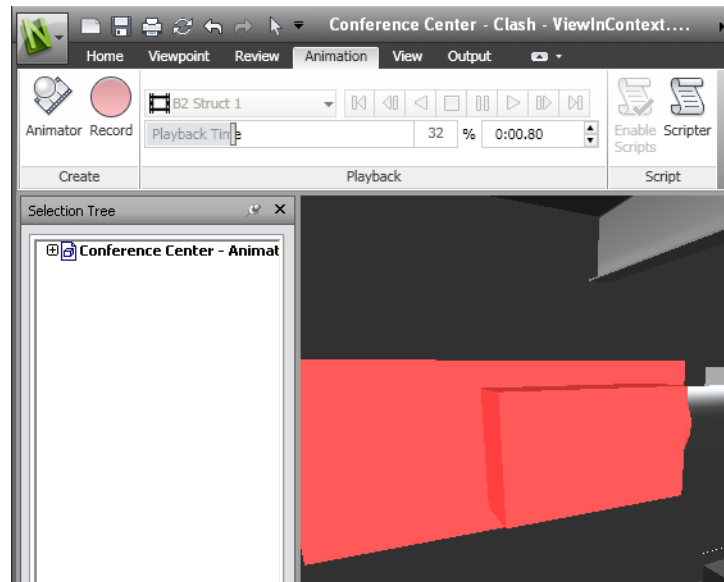


- 3 Select the **Simulation** check box in the **Display** area.
- 4 If the **TimeLiner** window is not already open, click **Home** tab ► **Tools** panel ► **TimeLiner** , and click the **Simulation** tab.
- 5 In the **Clash Detective** window, **Results** tab, select a clash in the **Results** list.

- 6 The simulation slider in the **TimeLiner** window is moved to the exact point at which the clash occurs. You can move the slider to investigate the events happening immediately before and after the clash.
- 7 Repeat this process to review all found clashes.

To review soft clash results

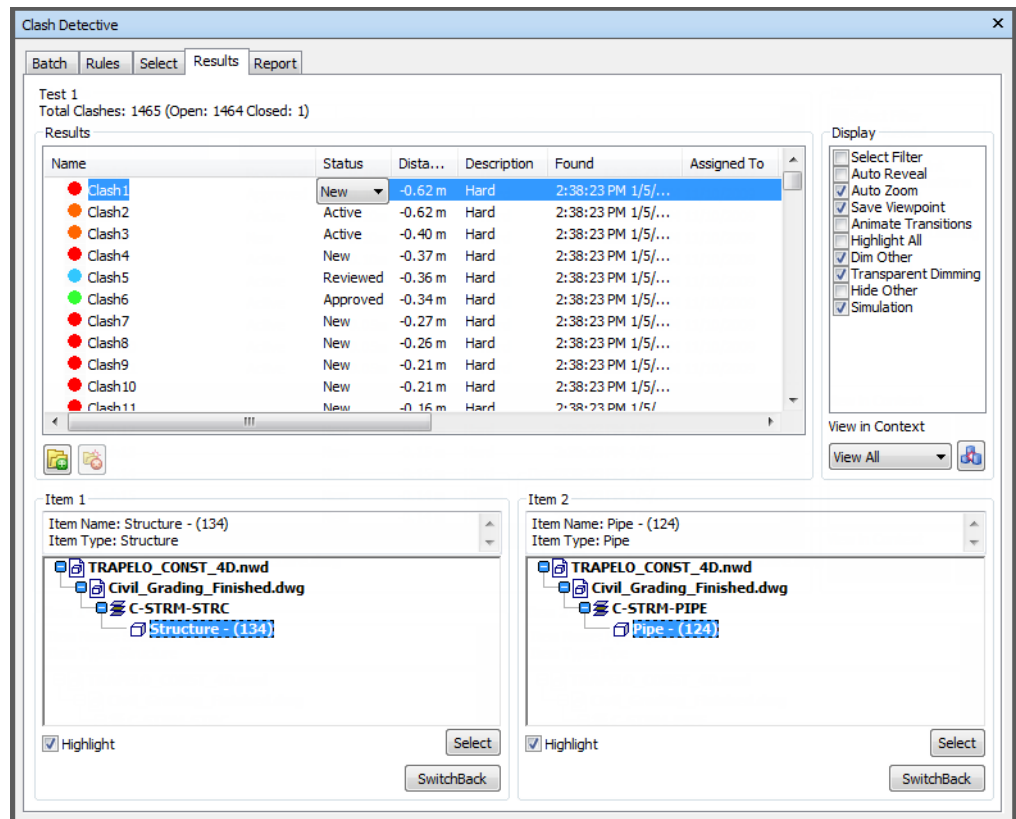
- 1 In the **Clash Detective** window, set up and run a soft clash test (see [Soft Clashing](#) (page 700)).
- 2 Click the **Results** tab.
- 3 Select the **Simulation** check box in the **Display** area.
- 4 In the **Clash Detective** window, **Results** tab, select a clash in the **Results** list.
- 5 Click the **Animation** tab on the ribbon. The **Playback Position** slider on the **Playback** panel is moved to the exact point at which the clash occurs. You can move the slider to investigate the events happening immediately before and after the clash.



- 6 Repeat this process to review all found clashes.

To review time-based soft clash results

- 1 In the **Clash Detective** window, set up and run a time-based soft clash test (see [Time-Based Soft Clashing](#) (page 701)).
- 2 Click the **Results** tab.



- 3 Select the **Simulation** check box in the **Display** area.
- 4 If the **TimeLiner** window is not already open, click **Home** tab ➤ **Tools** panel ➤ **TimeLiner** , and click the **Simulation** tab.
- 5 In the **Clash Detective** window, **Results** tab, select a clash in the **Results** list.
- 6 The simulation slider in the **TimeLiner** window is moved to the exact point at which the clash occurs. You can move the slider to investigate the events happening immediately before and after the clash.

- 7 Repeat this process to review all found clashes.

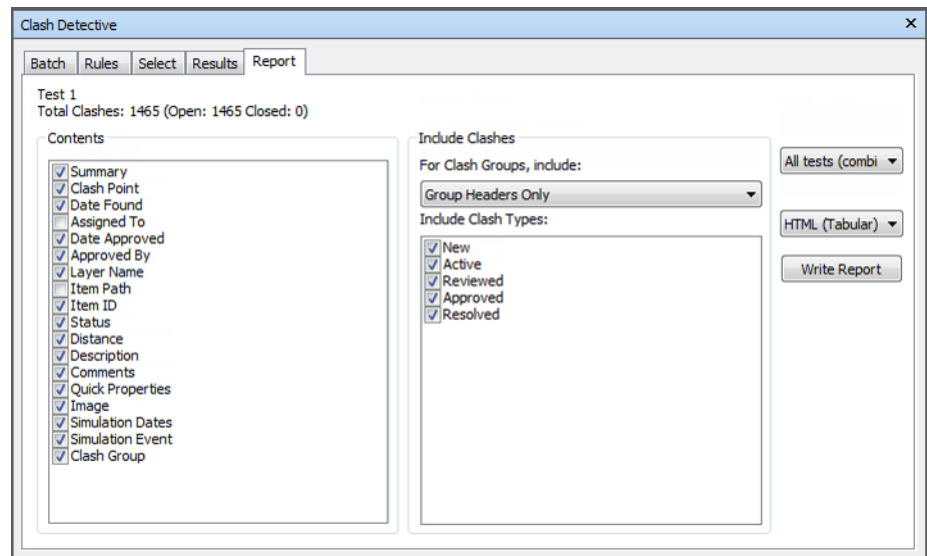
Report Clash Results

You can produce various **Clash Detective** reports. For example, reports can be used to communicate coordination issues to design teams that don't have access to Autodesk Navisworks.

For time-based clashes, it may be helpful to include in the report additional information about each static package in the clash. This information can be set up in the **Options Editor** by using the **Quick Properties** definitions.

To create a clash report

- 1 In the **Clash Detective** window, run the desired test (individually, or as a batch). If you run a batch test, on the **Batch** tab, select the test you want to view the results for.
- 2 Click the **Report** tab.



- 3 In the **For Clash Groups Include** box, **Include Clashes** area, specify how to display the clash groups in your report.

NOTE If your test does not contain any clash groups, this box is not available.

4 Choose from the following options:

- **Group Headers Only** - your report will only include summaries of the clash group folders that you have created.

	Name	Basement Clashes
	Distance*	-8.81m
	Status*	Active
	Clash Point*	1.22m, 7.97m, -0.18m
Item 1*		
ElementID	126987	
Layer	B2 BASEMENT	
Item Name*	Interior L(2-hr)-STUD SHAFT	
Item Type*	Solid	
Item 2*		
Item Name*	Layer0	
Item Type*	Face Set	

An example of an HTML report with group headers only

- **Individual Clashes Only** - your report will only contain individual clash results. For each clash that belongs to a group, an extra field, called **Clash Group**, can be added to the report to identify it. To enable this functionality, select the **Clash Group** check box in the **Contents** area.

	Name	Clash1
	Distance	-8.81m
	Clash Group	Basement Clashes
	Description	Hard
	Status	Active
	Clash Point	1.22m, 7.97m, -0.18m
	Date Created	2009/2/2 16:52:55
	Item 1	
Element ID	126987	
Layer	B2 BASEMENT	
Item Name	Interior L (2-hr)-STUD SHAFT	
Item Type	Solid	
Item 2		
Item Name	Layer0	
Item Type	Face Set	

An example of an HTML report with individual clashes only

- **Everything** - your report will contain both summaries of the clash group folders that you have created and individual clash results. For each clash that belongs to a group, an extra field, called **Clash Group**, can be added to the report to identify it. To enable this

functionality, select the **Clash Group** check box in the **Contents** area.

Name	Basement Clashes
Distance*	-8.81m
Status*	Active
Clash Point*	1.22m, 7.97m, -0.18m

Item 1*

Element ID	126987
Layer	B2 BASEMENT
Item Name*	Interior L (2-hr)-STUD SHAFT
Item Type*	Solid

Item 2*

Item Name*	Layer0
Item Type*	Face Set

Name	Clash1
Distance	-8.81m
Clash Group	Basement Clashes
Description	Hard
Status	Active
Clash Point	1.22m, 7.97m, -0.18m
Date Created	2009/2/2 16:52:55

Item 1

Element ID	126987
Layer	B2 BASEMENT
Item Name	Interior L (2-hr)-STUD SHAFT
Item Type	Solid

Item 2

Item Name	Layer0
Item Type	Face Set

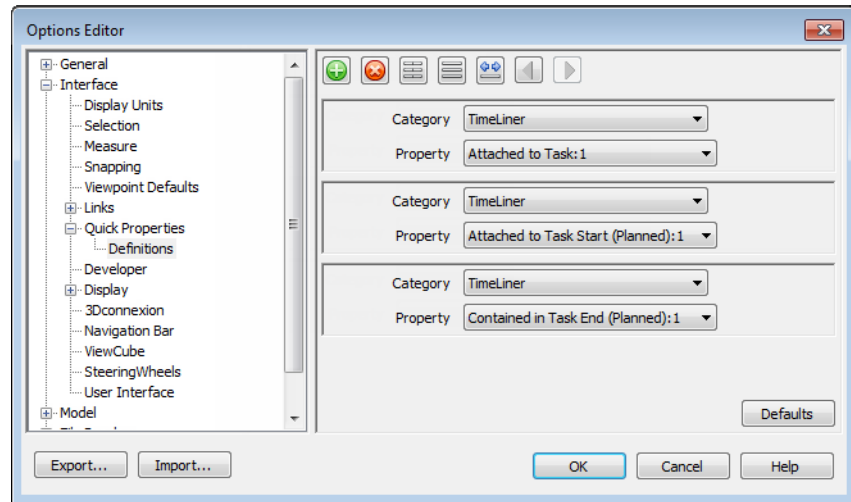
An example of an HTML report with both group headers and individual clashes

- 5 Use the **Include Clash Types** box to select the clash results that you want to report on.
- 6 In the **Contents** area select the check boxes for the data you want to appear in the report for each clash result. This can include **Quick Properties** relating to the items involved in the clash, how to find them in the standard **Selection Tree** from root to geometry, whether images or simulation information should be included, and so on.
- 7 Select the type of report in the **Report Type** box:
 - **Current Test** creates a single report file for the current test.
 - **All Tests (Combined)** creates a single file containing all results from all tests.

- **All Tests (Separate)** creates a separate file for each test containing all results.
- 8 Select the format of the report in the **Report Format** box:
- **XML** creates an XML file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the XML file.
 - **HTML** will create an HTML file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the HTML file.
 - **HTML (Tabular)** will create an HTML tabular file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the HTML file. HTML tabular files can be opened and edited in Microsoft Excel 2007 onwards.
- NOTE** To customize the appearance or layout of the HTML / HTML (Tabular) file, you will need to edit the `clash_report_html.xml` / `clash_report_html_tabular.xml` file. The installed file is located in the stylesheets subdirectory of the Autodesk Navisworks install directory. You can copy the edited file to the stylesheets subdirectory of any of the Autodesk Navisworks search directories. See [Search Directories](#) (page 159) for more information.
- **Text** creates a TXT file containing all the clash details and the location of a jpeg of each clash. On choosing this option, you will need to select or create a folder for the files and enter a name for the TXT file.
 - **As Viewpoints** creates a folder in the **Saved Viewpoints** dockable window, called the name of the test. Each clash is saved as a viewpoint in this folder, with a comment attached containing the clash result details.
- 9 Click the **Write Report** button to write the report.

Configure Quick Properties for your report

- 1 Click the application button  ➤ **Options**.
- 2 Expand the **Interface** node in the **Options Editor**, expand the **Quick Properties** option, and click **Definitions**.



- 3 Select **TimeLiner** in the **Category** box.
- 4 Select **Attach to Task: 1** in the **Property** box.
- 5 Add several more definitions with **TimeLiner** selected in the **Category** box, and the desired properties in the **Property** box.
- 6 Click **OK** to save the changes.

When you now select the **Quick Properties** check box in the **Contents** area on the **Report** tab of the **Clash Detective** window, your report will include the additional data you've specified here.

Use the Autodesk Vault Add-In

16

The Autodesk Vault add-in enables you to perform common vault functions on Autodesk Navisworks files (.nwc, .nwd, .nwf).

About the Autodesk Vault Add-In

A vault is a repository where documents and files are stored and managed. Autodesk Vault is a data management system that offers file security, version control, and multi-user support. Autodesk Vault add-in is available with Autodesk Navisworks Manage and Autodesk Navisworks Simulate, and supports connection to Autodesk Vault, Autodesk Vault Workgroup, Autodesk Vault Collaboration, and Autodesk Vault Professional.

The Autodesk Vault add-in enables you to perform common vault functions on Autodesk Navisworks files (.nwc, .nwd, .nwf). By default, the Vault add-in is not shown on the Autodesk Navisworks interface. To control the display of the Vault add-in, use the **Vault** setting in the **Options Editor**.

To display the Vault add-in on the Navisworks interface

- 1 Click the application button  ➤ **Options**.
- 2 In the **Options Editor**, expand the **Tools** node, and click **Vault**.
- 3 Select the **Show in User Interface** check box.
- 4 Click **OK**.
- 5 Restart the program. The Vault add-in is now available on the application menu and the ribbon.

Launching the Vault Application

You can start the Autodesk Vault application from within Navisworks Manage for vault administration and file maintenance. You need to have at least one Autodesk Vault product installed on your machine to do this.

Multiple Vault products can be installed on your machine at the same time. If you are logged into a vault server, launching the Vault application will open the Vault Explorer for the standalone vault client that matches the current log in information. If you are not logged into a vault server, it will open the Vault Explorer for the highest-level vault product that you have installed.

To launch Autodesk Vault

- 1 In Autodesk Navisworks application, click **Vault** tab ➤ **Access** panel
➤ **Go to Vault**  . The Vault Explorer opens, enabling you to browse the data in the vault.

Log into a Vault

Working with a vault allows your files to be safely stored and tracks versions of your files.

To work with files that are stored in the vault, you must log into the Autodesk Vault Server. If a vault has not been set up, contact your vault administrator. To ensure a secure working environment, it is important to have unique user accounts.

To log into a vault

- 1 Click **Vault** tab ➤ **Access** panel ➤ **Log In** .
- 2 In the **Log In** dialog box, enter the user name and password assigned to you by your vault administrator.
- 3 Enter the name of the computer where the server is installed, for example, `SERVERNAME`. If you do not know the name of the server, contact your administrator.
- 4 Enter the name of the vault database you want to log into.
- 5 Select the **Automatically Log in Next Session** check box to sign in automatically next time.

- 6 If you have a Microsoft Windows Active Directory account, you can log into the Autodesk Vault Server using your Microsoft Windows user account credentials. To do this, select the **Windows Authentication** check box.

NOTE This option applies only to Autodesk Vault Collaboration and Autodesk Vault Professional editions.

- Windows Authentication is available only for Active Directory domains.
- If you are not using Windows Authentication, then you must have a valid Autodesk Vault Server account.

NOTE If Windows Authentication is turned on, the **User Name** box displays the domain name combined with the Windows user name. Neither the user name nor the password can be changed.

- 7 Click **OK**.

Log out of a Vault

When you finish working with a vault, log out of the Autodesk data management server.

The **Log Out**  command is only available on the ribbon when you are logged in.

To log out of a vault

- 1 Click **Vault** tab ► **Access** panel ► **Log Out** .
- 2 Click **Close**.

Understanding the Working Folder

The working folder is the local directory where the files on which you can work on are stored.

Vault is a server/client data management solution which provides a centralized data storage. This centralized data storage mechanism enables you to access the same data during the whole design process. When a file is checked out or

when you open a file from a vault, the file is copied from the vault to a local folder called working folder. A working folder is defined by default, allowing you to begin working with a vault right away. The default working folder is:

My Documents\Vault

This means that each user has a unique working folder, isolated from other users. However, depending on your design requirements, you can specify a working folder that is located on a shared network drive, creating a shared working folder that all users access. In a multi-user environment, a shared working folder keeps all the working files consistently located in the same working folder. This prevents the lag between the upload to the server by one user and the subsequent download to all other users.

To use a shared working folder, the system administrator sets up and enforces a working folder on a shared network location for all project members to use.

Rules for using a shared working folder

Using a shared working folder means that multiple users may be working on the same files concurrently. To protect the group's data, all users **must** log into the vault before performing any file operations.

NOTE If a user does not log in, there is no way to verify that a file is checked out and the data is at risk of being overwritten.

The following guidelines will help prevent you from overwriting the changes made by someone else.

- 1 If a file is currently checked out to another user, you cannot perform the following operations:
 - Check Out
 - Undo Check Out

- 2 You cannot check out a file that is currently opened and checked out by another user.

NOTE You can check out a file that is opened as read-only by another user.

- 3 You can open a file as read-only when it is currently checked out to someone else using the same working folder.

To set up a shared working folder

NOTE You must be assigned the role of Administrator to perform this action.

- 1 Launch Autodesk Vault and log in as an administrator.
- 2 Select **Tools ► Administration**.
- 3 In the **Administration** dialog box, click the **Files** tab.
- 4 In the **Working Folder** section, click **Define**.
- 5 Select **Enforce a Consistent Working Folder for All Clients**.
- 6 Specify the shared network location for the working folder in the **Client Working Folder** box or browse for a location.

TIP The path to the shared working folder can be:

- a network path: \\designco\users\keyg
- a path containing a system variable for a directory:
\\designco\users\%username%

- 7 Click OK.
- 8 In the **Administration** dialog box, click **Close**.

For more information on enforcing the working folder, see the Autodesk Vault client help.

Check Out a File

You can check out the currently opened file. Checking out a file changes the read-only attribute of the local copy to read/write.

Once a vaulted file is checked out, it can be edited. Only one member of a team can check out a file at one time. No one else can modify a checked out file until it has been checked back into the vault. Use Autodesk Vault to see who has a file checked out. This information can be obtained from the User Name column.

To check a file out of a vault

- 1 Click the **Vault** tab ► **File Status** panel ► **Check Out**  .

Get Files from a Vault

You can use the open, append and merge commands to get files from the vault. Files are downloaded to the local working folder which is defined by the vault (for example, C:\Users\<USER>\My Documents\Vault in Windows 7).

The commands are located on the **Vault** tab, **Access** panel:

- **Open**  . Closes the file currently opened in Autodesk Navisworks and opens the file selected in the vault. You have a choice of opening the file as read-only, or checking it out for editing.
- **Append**  . Adds the selected files to the currently opened scene.
- **Merge**  . Merges the selected files into the currently opened scene. Any duplicate geometry and markup is automatically removed.

To open a vaulted file

- 1 Click **Vault** tab ► **Access** panel ► **Open**  .
- 2 In the **Select File** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to a vault location where the file you want to open is located.
- 3 Click the drop-down button next to the **Open** button, and click **Open (Check Out)** or **Open Read-Only** as appropriate. This closes the file currently opened in Autodesk Navisworks, and opens the file selected in the vault.

To append files from a vault into a current scene

- 1 Open an Autodesk Navisworks file.
- 2 Click **Vault** tab ► **Access** panel ► **Append**  .
- 3 In the **Select File** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to a vault location where the files you want to add are located.
- 4 Select the required files, and click **Open**.

TIP To select multiple files, use SHIFT and CTRL keys.

To merge files from a vault into a current scene

- 1 Open an Autodesk Navisworks file.
- 2 Click **Vault** tab ► **Access** panel ► **Merge** .
- 3 In the **Select File** dialog box, use the **Files of Type** box to select the appropriate file type, and navigate to a vault location where the files you want to add are located.
- 4 Select the required files, and click **Open**.

TIP To select multiple files, use SHIFT and CTRL keys.

Refresh a Vaulted File

You can retrieve the latest version of the currently opened file and its referenced files from a vault.

If the local version is older than the version in the vault, the latest version is copied from the vault to the local working folder, and the current file is reloaded. If the working copy of a file is newer than the most recent version of the file in the vault, you are prompted to choose either losing changes made to the current working copy or to not get the latest version of that file.

To refresh a vaulted file

- 1 Click **Vault** tab ► **File Status** panel ► **Refresh** . The opened file and its referenced files will be updated to the latest version in vault if appropriate.

Check a File into a Vault

You can use the **Check In** option on the ribbon to add files to the vault.

Use **Check In** when you are ready to check files into the vault for the first time. In addition, use **Check In** when you are done working on an Autodesk Navisworks file that has been checked out of the vault; check the file back into the vault to store the changes with the new version history. Include a comment with the file describing the changes made to this version of the data when checking it in. The latest version of a file in the vault is the last version

that was checked in. Once a file is checked in, the version in the vault is incremented and the latest changes are now available for others to check out.

Only one person can have a file checked out at one time. If you want to make a file currently checked out to you available to others without recording your changes, you can **Undo Check Out**. If no changes were made to a file that is checked back into the vault, the version history is not incremented. Use Autodesk Vault to see who has a file checked out. This information can be obtained from the User Name column.

To check a file into a vault

NOTE Before checking a file into the vault, save your changes.

- 1 Click **Vault** tab ► **File Status** panel ► **Check In** .
- 2 If the file has not been saved yet, you are prompted to save the file. Enter a name for the file, and click **Save**.
- 3 If you are checking the file into the vault for the first time, you are prompted to choose a location for the file in the vault. In the **Select Vault Location** dialog box, select a vault folder, or click the **New Folder** button to create a folder in the vault, and then click **OK**.
- 4 In the **Check In** dialog box:
 - If you want to keep the files checked out for further editing, select the **Keep Files Checked Out** check box.
 - If you want to remove the local copy once the file is checked in, click the **Close File and Delete Working Copy** check box.
- 5 Enter a comment in the text box describing the changes you made to this version of the file, if appropriate.
- 6 Click **OK**.

Undo File Check Out

Undo Check Out returns the opened checked out file to a checked in state in the vault without versioning it forward. Any changes made to the file are not stored in the vault.

NOTE If you choose the **Undo Check Out** command, you lose any changes you made to the local copy of the file.

To undo file check out

- 1 Click **Vault** tab ► **File Status** panel ► **Undo Check Out** .
- 2 If your local file has any unsaved edits, you are prompted to confirm that it will be overwritten with the data from vault. Click **OK**.
- 3 If your local file has already been saved, you are prompted to confirm that you want to overwrite new data with the older data from vault. Click **OK**.

Vault Settings

Log In Dialog Box

Use this dialog box to log into the vault database.

User Name	Enter the user name assigned to you by your system administrator.
Password	Enter the password assigned to you by your system administrator.
Server	Enter the name of the computer where the vault server is installed, for example, SERVERNAME. If you do not know the name of the vault server, contact your system administrator.
Database	Specifies the name of the vault database you want to log into. If you do not know the name of the vault database, contact your system administrator.
Automatically Log in Next Session	Sets the option to automatically log in to the specified vault without being prompted to enter User Name , Password , Server and Database .
Windows Authentication	Sets the option to log into the data management server using Microsoft Windows user account credentials. This disables the User Name and Password text boxes, and automatically fills in Windows domain log in account.

Ribbon: Vault tab ► **Access** panel ► **Log In** 

Check In Dialog Box

Use this dialog box to add your files to the vault.

Use the **Files Tree** tab to view files hierarchically within the vault. When the files are shown as a tree, you can see the structure of the files as they will be added to the vault.

The **Files Table** tab displays the files in the vault as a flat list.

Keep Files Checked Out	Keeps the files checked out after you check them into the vault. You can check in your changes so that others can get your latest changes.
Close Files and Delete Working Copies	Deletes the local copy of the file once it is checked in to the vault. If there are restrictions on a file that prevent it from being deleted, you are notified that the file cannot be deleted. NOTE Do not select the Close Files and Delete Working Copies check box when checking in files if you are using a shared working folder. Deleting the working copy from a shared working folder may delete files accessed by other users.
Settings	Opens the Settings dialog box, and enables you to organize the folder structure.
Enter Comments to Include with This Version	Text entered here is added to the Comment property and associated with this version of the file in the vault.

Ribbon: Vault tab ► **File Status** panel ► **Check In** 

Settings Dialog Box

Use this dialog box to organize your files in the vault.

Use Organized Folder Structure	Places all files and folders under a single folder. The original folder structure is retained to the extent that all referenced files exist at the same level or beneath the folder the host file resides in. Files that are not located on the same level or above the host file are placed under the host file automatically. This is the default option.
Place All Files in One Folder	Flattens the entire folder structure and all files are placed at the same level as the top-level host in the vault. The flatten command will not proceed when two or more file names are the same inside the same folder.
Preserve Locations if in Working Folder	Maintains the file structure of files being checked in from the local working folder.

TIP These vault settings help ensure your data is organized in a way that is suitable for use with the vault. As best practice, use organized folder structure and preserve the file locations.

Pointing device: Check In dialog box ➤ **Settings**

Select Vault Location Dialog Box

Use this dialog to specify a folder to check files into the vault for the first time.

NOTE You must save a file before you can select a vault location and check in a file.

Select a Vault Folder	Displays the current folder structure. Highlight an existing folder to check in files to the vault or create a new folder.
New Folder	Opens the Create Folder dialog box. You can also create folders in Autodesk Vault.

OK	Completes the select folder operation. The Check In dialog box is displayed.
Cancel	Cancels the select folder operation and closes the dialog box.

Ribbon: **Vault** tab ► **File Status** panel ► **Check In** 

Create Folder Dialog Box

Use this dialog box to create a new folder under the currently selected directory in the **Select Vault Location** dialog box.

New Folder	Enter the name for the folder that will be created in the vault. The folder name must be unique within the currently selected directory.
Library	Select this check box if you want the new folder to be a library folder.

Pointing device: **Select Vault Location** dialog box ► **New Folder**

Select File Dialog Box

Use this dialog box to get files from the vault.

Look In	Browse to the desired location in the vault.
File List section	A list of files in the chosen vault folder. Click a file to select it. When you merge or append files you can select multiple files by using SHIFT and CTRL keys.
File Name	Name of the file that you want to get from the vault.
Files of Type	Selects the type of file that you want to get from the vault.

Open

Once you selected the file, click the Open button. If you can see the drop-down button next to the Open button you can choose to open your file as read-only or to check it out for edit. This option is not available when you merge or append files.

Ribbon: Vault tab ➤ **Access** panel ➤ **Open** 

Ribbon: Vault tab ➤ **Access** panel ➤ **Append** 

Ribbon: Vault tab ➤ **Access** panel ➤ **Merge** 

Animation Export Dialog Box

Use this dialog box to export an animation to an AVI file or a sequence of image files.

Source

Source Selects the source from which to export the animation. Choose from:

- **Current Animator Scene.** The currently selected object animation.
- **TimeLiner Simulation.** The currently selected **TimeLiner** sequence.
- **Current Animation.** The currently selected viewpoint animation.

Renderer

Renderer Selects the animation renderer. Choose from:

- **Presenter.** Use this option when you require the highest rendering quality.
- **OpenGL.** Quickly renders your animation; this option is also ideal for previewing animations.

Output

Format Select the output format. Choose from:

- **JPEG.** Exports a sequence of static images, which are taken from individual frames in the animation. Use the **Options** button to select **Compression** and **Smoothing** levels.
- **PNG.** Exports a sequence of static images, taken from individual frames in the animation. Use the **Options** button to select **Interlacing** and **Compression** levels.

- **Windows AVI.** Exports animation as a commonly readable AVI file. Use the **Options** button to select the video compressor from the drop-down list, and adjust the output settings.

NOTE If a video compressor is not available on your computer, the **Configure** button may be unavailable.

- **Windows Bitmap.** Exports a sequence of static images, which are taken from individual frames in the animation. There are no **Options** for this format.

Options Enables you to configure the options for the selected output format.

Size

Type Use the drop-down list to specify how to set the size of the exported animation.

TIP For animations, a much lower resolution can be used than for static images, for example, 640x480.

Choose from:

- **Explicit.** Gives you full control of the width and height (the dimensions are in pixels).
- **Use Aspect Ratio.** Enables you to specify the height. The width is automatically calculated from the aspect ratio of your current view.
- **Use View.** Uses the width and height of your current view.

Width Enables you to enter the width in pixels, when available.

Height Enables you to enter the height in pixels, when available.

NOTE There is a maximum size of 2048 x 2048 pixels, for Autodesk Navisworks OpenGL output.

Options

FPS Specifies the number of frames per second; this setting is relevant for AVI files.

NOTE The higher the FPS, the smoother the animation will be. However, using a high FPS will considerably increase the rendering time. It is, usually, acceptable to use between 10 to 15 FPS.

Anti-Aliasing This option applies to OpenGL renderer only. Anti-aliasing is used to smooth the edges of the exported images. Select the appropriate value from the drop-down list. The higher the number, the smoother the image, but the longer they take to export. **4x** is adequate for most situations.

Ribbon: **Output** tab ► **Visuals** panel ► **Animation** 

 **Menu:** Application button ► **Export** ► **Images &**

Animations ► **Animation** 

 **Menu:** Classic user interface: **File** ► **Export** ► **Animation**

Appearance Profiler Dialog Box

Use this dialog box to create and manage appearance profiles, and apply them to your model.

Selector Area

Use this area to define and test the object selection criteria that will be used by an appearance profile selector.

By Property Tab

Category Use this field to enter the property category to search on, for example 'Item', 'System Type', 'Material'.

Property Use this field to enter the property type to search on, for example 'Name', 'System Type', 'Material'.

Equals/Does Not Equal Use the drop-down list to select appropriate condition operator, for example 'Equals'. Enter the property value to search for in the field provided, for example 'Equipment'.

Test Selection Button Once you have defined the search criteria, click the **Test Selection** button. All objects that satisfy the criteria will be selected in the model.

By Set Tab

Sets List This is a list of all search and selection sets available in the currently open file. To choose a set, click on it.

Refresh Button You can add or remove selection and search sets in your model. To synchronize your changes with the sets list shown in the Appearance Profiler dialog, click the **Refresh** button.

Test Selection Click this button to test the set you chose. All objects that satisfy the criteria will be selected in the model.

Appearance Area

Use this area to configure the appearance overrides that will be associated with an appearance profile selector.

Color Use the color picker to choose the color that will be used to override appearance of the selected objects.

Transparency Use the slider to choose transparency level (0 to 100%) that will be used to override appearance of the selected objects. You can also type the value into the corresponding box. A higher value makes objects more transparent; a lower value makes objects more opaque.

Selector List

The Selector List shows all configured appearance profile selectors. You can update settings for each of the selectors in the list, but you cannot move the selectors around. To correct the order of selectors, you may need to delete them from the list and start again.

Buttons

Add Use the color picker to choose the color that will be used to override appearance of the selected objects.

Update You can modify the object selection criteria and the appearance settings for each of the selectors added to the Selector list. To save the changes, click the **Update** button.

Delete Clicking this button deletes the selected appearance selector in the list.

Delete All Clicking this button deletes all appearance selectors in the list. You will be prompted to confirm your decision.

Load Enables you to open and use a previously saved appearance profile.

Save Enables you to save the current appearance profile with the .dat file extension.

Run Sequentially applies all selectors in the current appearance profile to your model.

Ribbon: Home tab ► Tools panel ► Appearance Profiler 

Assign Clash Dialog Box

Use this dialog box to assign a clash or clash group to a person/trade.

NOTE This dialog box is only available for 3D models.

Assign To Enter the name of the person/trade the clash, clash group or multiple clashes are to be assigned to.

NOTE If the clash is already assigned then the assignee will be displayed when the dialog box is opened.

Notes Enter any notes as required. These can be viewed in the [Comments Window](#) (page 400).

NOTE Previously entered notes are not displayed when the dialog box is opened.

Shortcut menu: Right-click a clash, multiple clashes or a clash group in the Results tab, and click **Assign** on the shortcut menu.

Background Settings Dialog Box

Use this dialog box to choose a background effect to use in the **Scene View**.

Mode Selects the type of background effect. Choose from:

- Plain
- Graduated
- Horizon

NOTE Horizon mode and the associated colors are only available for 3D models.

Color Sets the color for a plain background.

Top Color Sets the top color in a graduated background.

Bottom Color Sets the bottom color in the graduated background.

Sky Color Sets the sky color (top) in a horizon background. This option is available for 3D models only.

Horizon Sky Color Sets the sky color (bottom) in a horizon background. This option is available for 3D models only.

Horizon Ground Color Sets the ground color (top) in a horizon background. This option is available for 3D models only.

Ground Color Sets the ground color (bottom) in a horizon background. This option is available for 3D models only.

Ribbon: **View** tab ► **Scene View** panel ► **Background** 

Shortcut menu: Right-click a blank area in the scene, and click **Background** on the shortcut menu.

 **Menu:** Classic user interface: **Tools** ► **Background**

Collision Dialog Box

Use this dialog box to adjust the collision settings for the selected viewpoint in a 3D workspace.

By default, **Collision**, **Gravity**, **Auto Crouch**, and **Third Person** view are switched off.

NOTE This dialog box is only available for 3D models.

Collision Select this check box to define a viewer as a collision volume in **Walk** and **Fly** modes. As a result, a viewer acquires some mass, and cannot pass through other objects, points, or lines in the **Scene View**.

NOTE Selecting this check box changes the rendering prioritization so that objects around the viewer are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius and speed of movement.

Gravity Select this check box to give a viewer some weight in **Walk** mode. This option works in conjunction with **Collision**.

Auto Crouch Select this check box to enable a viewer to crouch under objects that are too low to pass under in **Walk** mode. This option works in conjunction with **Collision**.

Viewer

Radius Specifies the radius of the collision volume.

Height Specifies the height of the collision volume.

Eye Offset Specifies the distance below the top of the collision volume, where the camera will focus upon if **Auto Zoom** check box is selected.

Third Person

Enable Select this check box to use **Third Person** view. In **Third Person** view, an avatar is shown in the **Scene View** to represent the viewer.

Selecting this check box changes rendering prioritization so that objects around the avatar are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius, speed of movement, and the distance of the camera behind the avatar.

Auto Zoom Select this check box to automatically switch from **Third Person** view to first person view whenever the line of vision becomes obscured by an item.

Avatar Specifies the avatar that is used in **Third Person** view.

Angle Specifies the angle at which the camera looks at the avatar.

For example, 0° positions the camera directly behind the avatar; 15° makes the camera look down on the avatar at a 15° angle.

Distance Specifies the distance between the camera and the avatar.

TIP If you want to restore the default values, click the **Defaults** button.

Pointing device: **Edit Viewpoint** dialog box ➤ **Settings**

Convert Object Properties Dialog Box

Use this dialog box to select third-party applications for the DWG and DXF file reader to read object information from.

The DWG and DXF file reader can convert object information from a number of third-party applications that are built on AutoCAD.

Select the check boxes for all required applications.

Pointing device: **Options Editor** dialog box ➤ **File Readers** node
➤ **DWG/DXF** option ➤ **Advanced**

Culling Options Dialog Box

With the **AutoCAD File Exporter** plugin installed in an AutoCAD application, the **Culling Options** dialog box enables you to adjust geometry culling when you navigate a model in the **Navigator Palette**.

Area

Enable Specifies whether or not area culling is used.

Number of Pixels Below Which Objects Are Culled Specifies a value for the screen area in pixels below which objects are culled. For example, setting the value to 100 pixels means that any object within the model that would be drawn less than 10x10 pixels in size are discarded.

Backface

Turns on backface culling for all objects. Select from the following options:

- **Off.** Turns off backface culling.
- **Solid.** Turns on backface culling for solid objects only. This is the default option.
- **On.** Turns on backface culling for all objects.

Near Clip Plane

Furthest Distance Specifies the farthest distance between the camera and the **Near Clip Plane** position.

Automatically Generated Specifies whether or not Autodesk Navisworks automatically controls the **Near Clip Plane** position to give you the best view of the model.

Far Clip Plane

Closest Distance Specifies the closest distance between the camera and the **Far Clip Plane** position.

Automatically Generated Specifies whether or not Autodesk Navisworks automatically controls the **Far Clip Plane** position to give you the best view of the model.

TIP If you want to restore the default values, click the **Defaults** button.

Shortcut menu: Classic user interface: Right-click in the **Navigator Palette**, and click **Options ► Culling Options** on the shortcut menu.

Customize Dialog Box

Use this dialog box to customize the appearance and contents of the Autodesk Navisworks toolbars in the **Classic** user interface.

 **Menu:** Classic user interface: **Tools > Customize**

Shortcut menu: Classic user interface: right-click any toolbar, and click **Customize** on the shortcut menu.

Toolbars Tab

Use this tab to specify which toolbars should appear and which should be hidden, and to add and manage your own toolbars in the **Classic** user interface.

Toolbars Select the check box to display the corresponding toolbar. Clear the check box to hide the corresponding toolbar.

Buttons

New Displays the **New Toolbar** dialog box and creates a new custom toolbar. By default, new toolbars are named “Custom X” where “X” is the next available number added to the list.

Rename Displays the **Rename Toolbar** dialog box and enables you to rename a custom toolbar.

Delete Enables you to delete the selected custom toolbar.

Commands Tab

Use this tab to add and remove commands from the **Menu** bar and the toolbars in the **Classic** user interface.

NOTE You cannot add and remove commands from shortcut menus.

Categories Contains the groups of commands that can be customized.

Commands Contains all available commands for the selected category.

Selected Command

Description Activates when you've dropped the command onto a toolbar or a menu. Clicking this button gives you more information about the command.

Modify Selection Activates when you've dropped the command onto a toolbar or a menu. Clicking this button enables you to edit the command's appearance.

- **Reset.** Resets the selected command to its default appearance.
- **Delete.** Deletes the selected command.
- **Name.** Specifies the name of the command. Place an & (ampersand) in front of the letter you want to use as a keyboard shortcut. Pressing ALT together with this letter activates the command.
- **Default Style.** Uses the default style for the command.
- **Text Only.** The command is shown as text (as it's entered in the **Name** box).
- **Image and Text.** The command is shown as both text and image.
- **Begin a Group.** Adds a separator to the left of the command if it's added to a toolbar, or above the command, if it's added to a menu.

Options Tab

Use this tab to personalize appearance of toolbars and menus in the **Classic** user interface.

Personalized Menus and Toolbars

Always Show Full Menus Indicates whether the **Menu** bar contains all commands or not. By default, this check box is selected.

If you want to show only basic and frequently used commands on short versions of the menus, clear this check box.

Show Full Menus After a Short Delay Activates when the **Always Show Full Menus** check box is clear. Select this check box if you want to show all commands on the menu after a brief delay when you rest the mouse pointer on an open menu.

Reset Menu and Toolbar Usage Data Resets any changes you've made to the appearance of toolbars and menus to default settings.

Other

Large Icons Toggles the size of the toolbar buttons. This does not affect icons used on the menus.

List Font Names in Their Font Toggles the way the font names are shown in font lists.

Show ScreenTips on Toolbars Toggles the display of tooltips when the mouse pointer is over a toolbar button.

Show Shortcut Keys in ScreenTips Activates when the **Show ScreenTips on Toolbars** check box is selected. Toggles the display of shortcut keys in tooltips.

Menu Animations Defines the way menus are animated.

Default Collision Dialog Box

Use this dialog box to specify and save your preferred collision settings in a 3D workspace.

By default, **Collision**, **Gravity**, **Auto Crouch**, and **Third Person** view are switched off. When you modify default collision settings, your changes do not affect the currently opened Autodesk Navisworks file. They are used as soon as you open a new Autodesk Navisworks file, or start a new Autodesk Navisworks session.

Collision Select this check box to define a viewer as a collision volume in **Walk** and **Fly** modes. As a result, a viewer acquires some mass, and cannot pass through other objects, points, or lines in the **Scene View**.

NOTE Selecting this check box changes the rendering prioritization so that objects around the viewer are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius and speed of movement.

Gravity Select this check box to give a viewer some weight in **Walk** mode. This option works in conjunction with **Collision**.

Auto Crouch Select this check box to enable a viewer to crouch under objects that are too low to pass under in **Walk** mode. This option works in conjunction with **Collision**.

Viewer

Radius Specifies the radius of the collision volume.

Height Specifies the height of the collision volume.

Eye Offset Specifies the distance below the top of the collision volume, where the camera will focus upon if **Auto Zoom** check box is selected.

Third Person

Enable Select this check box to use **Third Person** view. In **Third Person** view, an avatar is shown in the **Scene View** to represent the viewer.

Selecting this check box changes rendering prioritization so that objects around the avatar are displayed with much higher detail than normal. The size of the region of high detail is based on collision volume radius, speed of movement, and the distance of the camera behind the avatar.

Auto Zoom Select this check box to automatically switch from **Third Person** view to first person view whenever the line of vision becomes obscured by an item.

Avatar Specifies the avatar that is used in **Third Person** view.

Angle Specifies the angle at which the camera looks at the avatar.

For example, 0° positions the camera directly behind the avatar; 15° makes the camera look down on the avatar at a 15° angle.

Distance Specifies the distance between the camera and the avatar.

TIP If you want to restore the default values, click the **Defaults** button.

Pointing device: Options Editor dialog box ➤ **Interface** node
➤ **Viewpoint Defaults** page ➤ **Settings**

Edit Key Frame Dialog Box

Use this dialog box to edit keyframes for animation sets, cameras, and section planes.

The fields in this dialog box are context-sensitive, depending on the animation type, and enable you to edit the animation operations the selected keyframe represents.

NOTE Animation sets and section planes are only available for 3D models.

Edit Key Frame - Animation Set

All values are relative to the previous keyframe, or, if it is the first keyframe, relative to the model's starting position.

Time Enables you to reposition the time of this keyframe. The value is the number of seconds since the start of the scene; it is not relative to the last keyframe.

Translate Enter the **X**, **Y**, and **Z** coordinate values to move the selected objects into this position.

Center Enter the **X**, **Y**, and **Z** coordinate values to move the center point of the rotation or scale into this position.

Rotate Enter the degrees of rotation around **X**, **Y**, and **Z** axis to move the selected objects into this position.

Orientation Enter the degrees of rotation around **X**, **Y**, and **Z** axis to modify the orientation of the rotation.

NOTE 0,0,0 represents world up.

Scale Enter the scaling factor around **X**, **Y**, and **Z** axis; 1 being the current size, 0.5 half, 2 being double and so on.

Color This check box indicates whether the color change is recorded in the keyframe or not. Enter the **R** (red), **G** (green), and **B** (blue) values for the new color.

If you don't want to enter the **R**, **G**, **B** values manually, click  and select the desired color.

Transparency This check box indicates whether the transparency change is recorded in the keyframe or not.

Enter the **%** value to adjust the transparency level (0 to 100%). A higher value makes the element more transparent; a lower value makes the element more opaque.

If you don't want to enter the transparency value manually, use the slider  to adjust the transparency level.

Interpolate Determines whether or not Autodesk Navisworks automatically interpolates between the current and the last keyframes. This is the default option. When disabled, there is no gradual movement between the two keyframes; instead the animation instantly jumps to the position/view of the second keyframe when it's reached. Also, there will be no colored animation bar between the keyframes.

Edit Key Frame - Camera

Time Enables you to reposition the time of this keyframe. The value is the number of seconds since the start of the scene; it is not relative to the last keyframe.

Position Enter the **X**, **Y**, and **Z** coordinate values to move the camera into this position.

Look At Enter the **X**, **Y**, and **Z** coordinate values to change the focal point for the camera.

Vertical Field of View, Horizontal Field of View Defines the area of the scene that can be viewed through the camera. You can adjust the values for both vertical and horizontal angles of view. A larger value produces a wider angle of view and a smaller value produces a narrower, or more tightly focused, angle of view.

NOTE When you modify the **Vertical Field of View**, the **Horizontal Field of View** is automatically adjusted, and vice versa to match the aspect ratio in Autodesk Navisworks.

Roll Rotates the camera around its front-to-back axis. A positive value rotates the camera counterclockwise, and a negative value rotates it clockwise.

Interpolate Determines whether or not Autodesk Navisworks automatically interpolates between the current and the last keyframes. This is the default option. When disabled, there is no gradual movement between the two keyframes; instead the animation instantly jumps to the position/view of the second keyframe when it's reached. Also, there will be no colored animation bar between the keyframes.

Edit Key Frame - Section Plane

Time Enables you to reposition the time of this keyframe. The value is the number of seconds since the start of the scene; it is not relative to the last keyframe.

Section Planes A list of current section planes.



Adds a section plane to the **Section Planes** list.



Deletes the section plane selected in the **Section Planes** list.

Distance The distance of the selected section plane across the model.

Plane Selects the vector that represents the angle of the section plane. The drop-down list contains a range of pre-defined values. If you select the **Define Custom** option, you can manually define the “up” vector for the plane.

Enabled Indicates whether the selected section plane is active or not.

Interpolate Determines whether or not Autodesk Navisworks automatically interpolates between the current and the last keyframes. This is the default option. When disabled, there is no gradual movement between the two keyframes; instead the animation instantly jumps to the position/view of the second keyframe when it’s reached. Also, there will be no colored animation bar between the keyframes.

Pointing device: Double-click a keyframe

Shortcut menu: Right-click a keyframe, and click **Edit** on the shortcut menu.

Edit Link Dialog Box

Use this dialog box to modify the selected database link.

For information about the settings on this dialog box, see the [New Link dialog box](#) (page 763).

Pointing device: **File Options** dialog box ► **DataTools** tab ► **Edit**

Edit Viewpoint Dialog Box

Use this dialog box to edit viewpoint attributes.

Camera

Position Enter the **X**, **Y**, and **Z** coordinate values to move the camera into this position. The Z coordinate values are not available in a 2D workspace.

Look At Enter the **X**, **Y**, and **Z** coordinate values to change the focal point for the camera. The Z coordinate values are not available in a 2D workspace.

Vertical Field of View, Horizontal Field of View Defines the area of the scene that can be viewed through the camera in a 3D workspace only. You can adjust the values for both vertical and horizontal angles of view.

A larger value produces a wider angle of view and a smaller value produces a narrower, or more tightly focused, angle of view.

NOTE When you modify the **Vertical Field of View**, the **Horizontal Field of View** is automatically adjusted, and vice versa to match the aspect ratio in Autodesk Navisworks.

Roll Rotates the camera around its front-to-back axis. A positive value rotates the camera counterclockwise, and a negative value rotates it clockwise.

NOTE This value is not editable when the viewpoint up vector stays upright (that is, when you use **Walk**, **Orbit** and **Constrained Orbit** navigation tools).

Motion

Linear Speed The speed of motion in a straight line for the viewpoint in a 3D workspace. The minimum value is 0 and the maximum is based on the size of the scene's bounding box.

Angular Speed The speed at which the camera turns in a 3D workspace.

Saved Attributes

This area applies to saved viewpoints only. If you are editing a current viewpoint, this area is greyed out.

Hide/Required Select this check box to save hidden/required markup information about objects in your model with the viewpoint. When you use a viewpoint again, the hidden/required markups set when the viewpoint was saved are reapplied.

NOTE Saving the state information with each viewpoint requires a relatively large amount of memory.

Override Material Select this check box to save material override information with the viewpoint. When you use a viewpoint again, the material overrides set when the viewpoint was saved are reapplied.

NOTE Saving the state information with each viewpoint requires a relatively large amount of memory.

Collision

Settings Opens the [Collision dialog box](#) (page 742). This functionality is available in a 3D workspace only.

Ribbon: Viewpoint tab ► **Save, Load & Playback** panel ► **Edit**

Current Viewpoint 

 **Menu:** Classic user interface: **Viewpoint** ► **Edit Current Viewpoint**

Export Rendered Image Dialog Box

Use this dialog box to export a rendered scene into one of the file types supported by Autodesk Navisworks.

NOTE This dialog box is only available for 3D models.

Type Use the drop-down list to select one of the file types supported by Autodesk Navisworks.

Choose from:

- **Targa**
- **TIFF**
- **JPEG**
- **Windows Bitmap**
- **EPix**
- **PostScript**
- **LWI**
- **HDR (High Dynamic Range Image)**
- **OpenEXR (High Dynamic Range Image)**
- **PNG**
- **QTVR Panoramic Movie**

NOTE The **QTVR Panoramic Movie** will effectively export 32 images, rotating about the current camera position to form a 360° panorama. Best results are achieved when the camera has zero tilt and is located in a position that has a 360° panoramic view, for example, in the center of a room.

■ **QTVR Object Movie**

NOTE The **QTVR Object Movie** will effectively export an extensive number of images, (based on number of pan frames x number of tilt frames, see [QTVR Object Movie Settings Dialog Box](#) (page 829)) pivoting the model about its center point. Best results are achieved when the model is relatively small or compact.

■ **Printer**

Browse Opens the **Save As** dialog box, and enables you to specify the location, and the name of the file you want to render to.

Size

Type Use the drop-down list to specify how to set the size of the exported image.

Choose from:

- **Explicit.** Gives you full control of the width and height (the dimensions are in pixels).
- **Use Aspect Ratio.** Enables you to specify the height. The width is automatically calculated from the aspect ratio of your current view.
- **Use View.** Uses the width and height of your current view.
- **Use Printer Page.** Sizes the image to the page setup size of the default printer.

Width Enables you to enter the width in pixels, when available.

Height Enables you to enter the height in pixels, when available.

Ribbon: **Output** tab ► **Visuals** panel ► **Rendered Image** 

 **Menu:** Application button ► **Export** ► **Images & Animations** ► **Rendered Image**

 **Menu:** Classic user interface: **File** ► **Export** ► **Rendered Image**

File Options Dialog Box

Use this dialog box to control the appearance of the model and the speed of navigation around it, and also to create and configure links to external databases.

NOTE Some of the tabs are only available when working with 3D models.

When you modify any of the options in this dialog box, your changes are saved in the currently opened Autodesk Navisworks file, and apply to this file only.

Ribbon: **Home** panel ► **Project** tab ► **File Options** 

 **Menu:** Classic user interface: **Tools** ► **File Options**.

Culling Tab

Use this tab to adjust geometry culling in the opened Autodesk Navisworks file.

NOTE The Clipping Planes and Backface options are only available for 3D models.

Area

Enable Specifies whether or not area culling is used.

Number of Pixels Below Which Objects Are Culled Specifies a value for the screen area in pixels below which objects are culled. For example, setting the value to 100 pixels means that any object within the model that would be drawn less than 10x10 pixels in size are discarded.

Clipping Planes

Near

Automatic Select this radio button to make Autodesk Navisworks automatically control the near clip plane position to give you the best view of the model. The **Distance** box becomes unavailable.

Constrained Select this radio button to constrain the near clipping plane to the value set in the **Distance** box.

Autodesk Navisworks uses the provided value unless doing so affects performance (for example, makes the whole model invisible), in which case it adjusts the near clip plane position as necessary.

Fixed Select this radio button to set the near clipping plane to the value provided in the **Distance** box.

Distance Specifies the farthest distance between the camera and the near clipping plane position in constrained mode.

Specifies the exact distance between the camera and the near clipping plane position in fixed mode.

NOTE Nothing is drawn between the camera and the near clipping plane; when you override automatic mode, make this value small enough to display your data. Also, overriding automatic mode with values under 1 can produce unpredictable results.

Far

Automatic Select this radio button to make Autodesk Navisworks automatically control the far clipping plane position to give you the best view of the model. The **Distance** box becomes unavailable.

Constrained Select this radio button to constrain the far clipping plane to the value set in the **Distance** box.

Autodesk Navisworks uses the provided value unless doing so affects performance (for example, makes the whole model invisible), in which case it adjusts the far clip plane position as necessary.

Fixed Select this radio button to set the far clipping plane to the value provided in the **Distance** box.

Distance Specifies the closest distance between the camera and the far clipping plane position in constrained mode.

Specifies the exact distance between the camera and the far clipping plane position in fixed mode.

NOTE Nothing is drawn beyond this plane; when you override automatic mode, make this value large enough to include your data. Additionally, using the ratio of the far clipping plane to near clipping plane in excess of 10000 can produce unwanted effects.

Backface

Turns on backface culling for all objects. Select from the following options:

- **Off.** Turns off backface culling.
- **Solid.** Turns on backface culling for solid objects only. This is the default option.
- **On.** Turns on backface culling for all objects.

TIP If you can see through some objects, or some object parts are missing, turn off backface culling.

TIP If you want to restore the default values, click the **Reset to Defaults** button.

Orientation Tab

Use this tab to adjust the real-world orientation of your model.

NOTE This tab is only available for 3D models.

Up

X, Y, Z Specify the **X**, **Y**, and **Z** coordinate values. By default, Autodesk Navisworks takes the positive Z-axis as Up.

North

X, Y, Z Specify the **X**, **Y**, and **Z** coordinate values. By default, Autodesk Navisworks takes the positive Y-axis as North.

TIP If you want to restore the default values, click the **Defaults** button.

Speed Tab

Use this tab to adjust the frame rate speed to reduce the amount of drop-out during navigation.

TIP If this does not improve navigation, try switching off the **Guarantee Frame Rate** option.

Frame Rate Specifies the number of frames per second (FPS) that are rendered in the **Scene View**.

The default setting is 6. You can set the frame rate from 1 through 60 frames per second. Reducing the value reduces drop-out, but can cause jerky movement during navigation. Increasing the value ensures a smoother navigation, but increases drop-out.

TIP If you want to restore the default values, click the **Defaults** button.

Headlight Tab

Use this tab to change the intensity of the scene's ambient light and headlight for Head Light mode.

NOTE This tab is only available for 3D models.

Ambient Use the slider to control the overall brightness of the scene.

Headlight Use the slider to control the brightness of the light located at the camera.

NOTE To see the effect your changes have on the model in the **Scene View**, apply **Head Light** mode in the ribbon.

Scene Lights Tab

Use this tab to change the intensity of the scene's ambient light for Scene Lights mode.

NOTE This tab is only available for 3D models.

Ambient Use the slider to control the overall brightness of the scene.

NOTE To see the effect your changes have on the model in the **Scene View**, apply **Scene Lights** mode in the ribbon.

DataTools Tab

Use this tab to create and manage links between the opened Autodesk Navisworks file and external databases.

DataTools Links Displays all database links in Autodesk Navisworks file. Select the check box next to the link to activate it.

IMPORTANT You cannot activate links with insufficient or invalid configuration information.

Buttons

New Opens the [New Link dialog box](#) (page 763) where you can specify the link parameters.

Edit Opens the [Edit Link dialog box](#) (page 751) where you can modify the parameters for the selected database link.

Delete Deletes the selected database link.

Import Lets you select and open previously saved datatools files.

Export Saves the selected database link as a datatools file.

NOTE If you want to restore the default values, click the **Defaults** button.

Units and Transform Dialog Box

Use this dialog box to adjust the units and file transforms for the 3D file selected in the **Selection Tree** or the currently opened 2D sheet.

NOTE You can only adjust units and transforms for one 3D file or 2D sheet at a time.

Model Units

Units Lets you choose the units for the loaded 3D model.

Sheet Units

Units Lets you choose the units for the loaded 2D sheet.

Origin

Origin Enter the **X**, **Y**, and **Z** coordinate values to move the file geometry into this position.

NOTE Z coordinate values are not available for 2D sheets.

Reflected Transform Select this check box if you use a negative scale for your transform.

NOTE **Reflected Transform** is not available for 2D sheets.

Rotation

Enter the rotation angle.

Select the rotation axis by entering values greater than 0. The model or sheet geometry rotates about its origin point. Positive numbers rotate the model or sheet counterclockwise; negative numbers rotate model or sheet clockwise.

Scale

Enter the scaling factor around **X**, **Y**, and **Z** axis for the file geometry; 1 being the current size, 0.5 half, 2 being double and so on. To scale proportionally, ensure the X, Y and Z scale values are equal. Entering negative values flips the file geometry inside out.

NOTE Negative scale values cannot be used on 2D sheets. Also Z coordinate values are not available for 2D sheets.

Shortcut menu: 3D model: Right-click the desired item in the **Selection Tree**, and click **Units and Transform** on the shortcut menu

Shortcut menu: 2D sheet: Right-click the sheet in the **Scene View**, and click **Units and Transform** on the shortcut menu

Image Export Dialog Box

Use this dialog box to export the current viewpoint into one of the file types supported by Autodesk Navisworks.

Output

Format Use the drop-down list to select one of the image types supported by Autodesk Navisworks.

Choose from:

- **JPEG**
- **PNG**
- **Windows Bitmap**

Options Enables you to specify the options for PNG and JPG files. For PNG images, you can select the **Interlacing** and **Compression Level** settings, and for the **JPEG** images you can select **Compression** and **Smoothing** levels.

Size

Type Use the drop-down list to specify how to set the size of the exported image.

Choose from:

- **Explicit.** Gives you full control of the width and height (the dimensions are in pixels).
- **Use Aspect Ratio.** Enables you to specify the height. The width is automatically calculated from the aspect ratio of your current view.
- **Use View.** Uses the width and height of your current view.

Width Enables you to enter the width in pixels, when available.

Height Enables you to enter the height in pixels, when available.

Options

Anti-Aliasing This option applies to OpenGL renderer only. Anti-aliasing is used to smooth the edges of the exported images. Select the appropriate value from the drop-down list. The higher the number, the smoother the image, but the longer they take to export. 4x is adequate for most situations.

Ribbon: **Output** tab ➤ **Visuals** panel ➤ **Image** 

 **Menu:** Application button ➤ **Export** ➤ **Images & Animations** ➤ **Image**

 **Menu:** Classic user interface: **File** ➤ **Export** ➤ **Image**

InfoCenter Settings Dialog Box

Use this dialog box to specify **InfoCenter** and **Communication Center** settings.

Buttons

OK Saves changes and closes the **InfoCenter Settings** dialog box.

Cancel Discards changes and closes the **InfoCenter Settings** dialog box.

Help Displays the context-sensitive help.

Pointing device: In the **InfoCenter** box, click the Subscription Center button/Communication Center button/Favorites button ➤ **InfoCenter Settings**  button

General Node

Use the **General** node to select your current location, frequency for checking new online content and option to turn on or off animated transition effects for the **InfoCenter** panels.

Please Indicate the Country/Region Nearest to Your Current Location Set the country in which Autodesk Navisworks users work. This is used for tailoring location-specific **Communication Center** content.

Check for New Online Content Specifies how often **Communication Center** checks for new content.

Use Animated Transition Effects for Panels Check to animate panel transitions.

Pointing device: InfoCenter Settings dialog box ► **General** node

Communication Center Node

Use the **Communication Center** node to set the maximum age of the articles displayed on the **Communication Center** panel.

Hide Results Which Are More Than X Days Old Select this check box to have **InfoCenter** hide search results older than the numeric value you specify.

CAD Manager Channel Used by other Autodesk products to specify the RSS feeds published by a CAD manager, the **CAD Manager Channel** is not enabled in Autodesk Navisworks.

Display Name Type the name to be displayed in the **Search Results** panel.

Pointing device: InfoCenter Settings dialog box ► **Communication Center** node

Autodesk Channels Page

Use the settings on this page to adjust the settings for **Communication Center**.

By default, all available channels are selected. You cannot add or remove channels from the grid, and you cannot edit data in the grid.

Select Channels to Display in the Communication Center Panel Select the channels and the number of articles you want to display in the **Communication Center** panel.

Pointing device: InfoCenter Settings dialog box ► **Communication Center** node ► **Autodesk Channels** page

Balloon Notification Page

Use the settings on this page to adjust balloon notifications.

Enable Balloon Notification for These Sources Select this check box to enable balloon notifications in the product. Balloon notifications appear over the **InfoCenter** box when any new information is available from the selected sources.

Live Update Channel (New Software Updates) Select this check box to receive balloon notification of available software updates.

Product Support Information Channel Select this check box to receive balloon notification of new product information.

CAD Manager Channel The **CAD Manager Channel** is not enabled in Autodesk Navisworks.

RSS Feeds Select this check box to receive balloon notification of new RSS feeds.

Number of Seconds Balloon Notification Displays Enter a numeric value to indicate the amount of time to display balloon notifications.

% Transparency of Balloon Notification Enter a numeric value to indicate the transparency of balloon notifications.

Alternatively, drag the slider toward **Opaque** to decrease the balloon notification transparency percentage or toward **Transparent** to increase the transparency percentage.

Pointing device: InfoCenter Settings dialog box ► **Communication Center** node ► **Balloon Notification** page

RSS Feeds Page

Use the settings on this page to RSS feeds.

RSS Subscription Add. Specify the path for the RSS feed you want to add. After the RSS feed has been added to the **RSS Subscription** list, under **Items to Display** enter a numeric value to indicate the number of items to display.

Remove. Remove a selected RSS feed from the **RSS Subscription** list.

Pointing device: InfoCenter Settings dialog box ► **Communication Center** node ► **RSS Feeds** page

New Link Dialog Box

Use this dialog box to add a database link to the opened Autodesk Navisworks file.

Name Specifies the name of the database link. This name appears as a tab on the **Properties** control bar. Each link name must be unique.

Connection

ODBC Driver Defines the type of database for the link. Use the drop-down list to select the appropriate ODBC Driver.

Setup Opens the **Database Setup** wizard. Use it to enter the connection options. If you have problems setting up your connection details, contact your database administrator.

When you finish, the box underneath shows the connection string.

Hold Open for Application Lifetime Specifies whether or not the database link is open until you exit Autodesk Navisworks.

SQL String

Defines which table in the database to query. If you are not familiar with SQL, you may need to ask your database administrator to set the SQL statement for you.

Fields

Defines which columns to display as link categories on the **Properties** control bar.

Field Name Specifies the name of the column in the selected database table. You must enter the exact name.

Display Name Specifies the link category name shown on the **Properties** control bar. This name is automatically completed for you, but you can modify it, if necessary.

Pointing device: **File Options** dialog box ► **DataTools** tab ► **New**

Options Editor Dialog Box

Use the Options Editor to adjust program settings for Autodesk Navisworks sessions.

The settings that you set up in the **Options Editor** are persistent across all Autodesk Navisworks sessions. You also share the modified settings with other members of your team.

The options are presented in a hierarchical tree structure. Clicking  expands the nodes, clicking  collapses the nodes.

Ribbon: Application button ► **Options**

 **Menu:** Classic user interface: **Tools > Global Options**

Shortcut menu: Scene **> Global Options**

Buttons

Export Displays the **Select Options to Export** dialog box, where you can select the global options you want to export (or “serialize”). If an option cannot be exported, it is unavailable.

Import Displays the **Open** dialog box, where you can browse to the file with the required global option settings.

OK Saves the changes, and closes the **Options Editor**.

Cancel Discards the changes, and closes the **Options Editor**.

Help Displays the context-sensitive help.

General Node

Use the settings in this node to adjust the buffer size, file locations, number of recent file shortcuts you want Autodesk Navisworks to store, and the auto-save options.

TIP If you want to restore the default values, click the **Defaults** button.

Pointing device: **Options Editor** dialog box **> General** node

Undo Page

Use the settings on this page to adjust the buffer size.

Pointing device: **Options Editor** dialog box **> General** node **> Undo** page

Buffer Size (KB) Specifies the amount of space Autodesk Navisworks allocates for saving undo/redo actions.

Locations Page

Use the options on this page to share global Autodesk Navisworks settings, workspaces, datatools, avatars, Clash Detective rules, Presenter archives, custom Clash Detective tests, object animation scripts, and so on, with other users.

The settings can be shared across an entire project site, or across a specific project group depending on the required level of granularity.

When you run Autodesk Navisworks for the first time, the settings are picked up from the installation directory. Subsequently, Autodesk Navisworks examines the current user profile and the all users profile on the local machine, and then checks the settings in the **Project Directory** and the **Site Directory**. The files in the **Project Directory** take precedence.

Project Directory Click  to open the **Browse for Folder** dialog box, and locate the directory that contains the Autodesk Navisworks settings specific to a project group.

Site Directory Click  to open the **Browse for Folder** dialog box, and locate the directory that contains the Autodesk Navisworks settings standard across the entire project site.

See also:

[Location Options](#) (page 154)

[How do I share the program settings on a site and project basis?](#) (page 73)

Environment Page

Use the settings on this page to adjust the number of recent file shortcuts stored by Autodesk Navisworks.

Maximum Recently Used Files Specifies how many file shortcuts Autodesk Navisworks can remember. By default, shortcuts to the four most recently opened files can be displayed.

Auto-Save Page

Use the setting on this page to adjust the auto-save options.

Enable Auto-Save Indicates whether Autodesk Navisworks automatically saves Autodesk Navisworks files. By default, this check box is selected.

Clear this check box if you do not want to save Autodesk Navisworks files automatically.

Auto-Save File Location Specifies how the backup files are saved. Select from the following options:

- Auto-Save to a Specific Directory - this is the default option.

Directory

The default directory for auto-save is: <USERPROFILE>\ *Application Data*\<PRODUCTFOLDER>\AutoSave

Click  to open the **Browse for Folder** dialog box, and select the desired location for auto-save.

Manage Disk Space

Indicates whether the size of the disk space restricts the creation of backup files. This check box is selected by default.

Purge Old Auto-Save Files When This Folder Exceeds (MB)

Active when the **Manage Disk Space** check box is selected. Specifies the maximum directory size for backup files. The default value is 512 MB.

When the size of the auto-save folder exceeds the specified value, Autodesk Navisworks deletes the oldest backup file (based on the date modified).

- Auto-Save Alongside the Current File

Frequency

Time Between Saves (Minutes) Defines the time interval between auto-saves of significant file changes.

By default, a backup file is saved every 15 minutes following a significant change to a Autodesk Navisworks file.

History

Maximum Previous Versions Determines how many backup files are stored. By default, it's three files. When the number of auto-saved files exceeds the specified value, Autodesk Navisworks deletes the oldest backup file (based on the date modified).

Interface Node

Use the settings in this node to customize Autodesk Navisworks interface.

TIP If you want to restore the default values, click the **Defaults** button.

Display Units Page

Use this page to customize the units used by Autodesk Navisworks.

Linear Units Use the drop-down list to select the desired linear value. **Meters** are used by default.

Angular Units Use the drop-down list to select the desired angular value. **Degrees** are used by default.

Decimal Places Specifies the number of decimal places used by units.

Fractional Display Precision Specifies the level of fraction used by units. This box is enabled for fractional units only.

Selection Page

Use the options on this page to configure the way geometry objects are selected, and highlighted.

Pick Radius Specifies the radius, in pixels, that an item has to be within in order for it to be selected.

Resolution Specifies the level of selection used by default.

When you click in the **Scene View**, Autodesk Navisworks requires a starting point for the object path in the **Selection Tree** box to identify the selected item. You can choose one of the following options:

- **Model** - the object path starts at the model node; as a result, all objects in the model are selected.
- **Layer** - the object path starts at the layer node; as a result all objects within a layer are selected.
- **First Object** - the object path starts at the highest level of objects below the layer node, if applicable.
- **Last Object** - the object path starts at the lowest level of objects in the **Selection Tree**. Autodesk Navisworks looks for composite objects first, and if none are found, the geometry level is used instead. This is the default option.
- **Last Unique** - the object path starts at the first unique level of objects (not multiple-instanced) in the **Selection Tree**.
- **Geometry** - the object path starts from the geometry level in the **Selection Tree**.

Compact Tree Specifies the level of detail shown on the **Compact** tab of the **Selection Tree**.

Use one of the following options:

- **Models** - the tree is restricted to displaying model files only.
- **Layers** - the tree can be expanded down to the layer level.
- **Objects** - can be expanded down to the objects level, but without the levels of instancing shown on the **Standard** tab.

Highlight

Enabled Indicates whether Autodesk Navisworks highlights the selected items in the **Scene View**.

Clear this check box if you don't want to highlight selected items.

Method Specifies how the objects are highlighted. Select one of the following options:

- Shaded
- Wireframe
- Tinted

Color Click  to specify the highlight color.

Tint Level (%) Use the slider to adjust the tint level.

Measure Page

Use the options on this page to adjust the appearance and style of the measure lines.

Line Thickness Specifies the thickness of the measure lines.

Color Click  to specify the color of the measure lines.

In 3D Select this check box to draw the measure lines in 3D.

If the measure lines become obscured by other geometry, clear this check box to draw the lines in 2D over the top of geometry.

Show Measurement Values in Scene View Select this check box if you want to display the dimension labels in the **Scene View**.

Use Center Lines When this check box is selected, the shortest distance measurements snap to the center lines of parametric objects.

When this check box is clear, the surface of the parametric objects is used for the shortest distance measurement instead.

NOTE Changing this option does not affect any measurement currently in place. To see any changes, clear the measurement, and start again.

Snapping Page

Use the options on this page to adjust the cursor snapping.

Picking

Snap to Vertex Select this check box to snap the cursor to the nearest vertex.

Snap to Edge Select this check box to snap the cursor to the nearest triangle edge.

Snap to Line Vertex Select this check box to snap the cursor to the nearest line end.

Tolerance Defines the snapping tolerance. The smaller the value, the closer the cursor must be to a feature in the model before it snaps to it.

Rotation

Angles Specifies the multiplier for the snapping angle.

Angle Sensitivity Defines the snapping tolerance. The value you enter here determines how close to the snapping angle the cursor must be for snap to take effect.

Viewpoint Defaults Page

Use the options on this page to define attributes that are saved with viewpoints when you create them.

When you modify default viewpoint settings, your changes do not affect the currently opened Autodesk Navisworks file. They are used as soon as you open a new Autodesk Navisworks file, or start a new Autodesk Navisworks session.

Save Hide/Required Attributes Select this check box to save viewpoints with hidden/required markup information about objects in your model. When you use a viewpoint again, the hidden/required markups set when the viewpoint was saved are reapplied.

By default, this check box is clear, as saving the state information with each viewpoint requires a relatively large amount of memory.

Override Material Select this check box to save viewpoints with material override information. When you use a viewpoint again, the material overrides set when the viewpoint was saved are reapplied.

By default, this check box is clear, as saving the state information with each viewpoint requires a relatively large amount of memory.

Override Linear Speed By default, the linear navigation speed is directly related to the size of your model. Select this check box, if you want to set a specific navigation speed manually. This option is used in a 3D workspace only.

Default Linear Speed Specifies the default linear speed value. This option is used in a 3D workspace only.

Default Angular Speed Specifies the default speed at which the camera turns. This option is used in a 3D workspace only.

Collision

Settings Opens the [Default Collision dialog box](#) (page 747) where you can adjust the collision, gravity, crouching and third person view settings.

Links Page

Use the options on this page to customize the way links are displayed in the **Scene View**.

TIP If you want to restore the default values, click the **Defaults** button.

Show Links Toggles the display of links in the Scene View.

In 3D Indicates whether the link icons are drawn in 3D in the **Scene View**. Select this box if you want the links to float in 3D space just in front of their attachment points to the geometry.

If the links become obscured by other geometry, clear this check box to draw the link icons in 2D over the top of geometry.

Max Icons Specifies the maximum number of icons to draw in the **Scene View**.

Hide Colliding Icons Select this check box to hide the link icons that appear overlapped in the **Scene View**.

Cull Radius Specifies how close to the camera links have to be before they are drawn in the **Scene View**. Any links further away than this distance are not drawn. The default value of 0 means that all links are drawn.

X Leader Offset, Y Leader Offset Links can be drawn with leader lines (arrows) pointing to the attachment point on the geometry that the link is attached to. Enter the X- and Y- values to specify the number of pixels to the right and up that these leader lines use.

Standard Categories Page

Use the settings on this page to switch the displaying of links based on their categories.

Hyperlink

Icon Type Specifies how to display this link category.

Select one of the following options:

- **Icon** - links are represented by default icons  and  in the **Scene View**.
- **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Label

Icon Type Specifies how to display this link category.

Select one of the following options:

- **Icon** - links are represented by default icons  and  in the **Scene View**.
- **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Clash Detective

Icon Type Specifies how to display this link category.

Select one of the following options:

■ **Icon** - links are represented by default icons in the **Scene View**:

-  - links to new clashes
-  - links to active clashes
-  - links to resolved clashes
-  - links to reviewed clashes
-  - links to approved clashes

■ **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Hide Icons Without Comments Select this check box to display only the links that have comments in the **Scene View**.

TimeLiner

Icon Type Specifies how to display this link category.

Select one of the following options:

■ **Icon** - links are represented by default icons in the **Scene View**:

-  - links to manually created tasks
-  - links to tasks with valid links
-  - links to tasks with broken links

■ **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Hide Icons Without Comments Select this check box to display only the links that have comments in the **Scene View**.

Viewpoints

Icon Type Specifies how to display this link category.

Select one of the following options:

■ **Icon** - links are represented by default icons in the **Scene View**:

-  - links to viewpoints saved in perspective mode
-  - links to viewpoints saved in orthographic mode

■ **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Hide Icons Without Comments Select this check box to display only the links that have comments in the **Scene View**.

Redline Tags

Icon Type Specifies how to display this link category.

Select one of the following options:

■ **Icon** - links are represented by default icons  in the **Scene View**.

■ **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Hide Icons Without Comments Select this check box to display only the links that have comments in the **Scene View**.

Sets

Icon Type Specifies how to display this link category.

Select one of the following options:

■ **Icon** - links are represented by default icons in the **Scene View**:

-  - links to selection sets
-  - links to search sets

■ **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Visible Select this check box to display this link category in the **Scene View**.

Hide Icons Without Comments Select this check box to display only the links that have attached comments in the **Scene View**.

User-Defined Categories Page

Use this page to view custom link categories.

The padlock  icon indicates that you cannot add or remove categories directly from here.

Buttons

Grid View Click  to display custom link categories in a tabular format.

List View Click  to display custom link categories in a list format (the same way as the standard link categories are shown).

Records View Click  to display link categories as records.

Previous and Next Element Use  and  to navigate between link categories. If you clicked the **Records View** button, this is the only way to move between the records.

Visible Select this check box to display the corresponding link category in the **Scene View**.

Icon Type Select one of the following options:

- **Icon** - links are represented by default icons  and  in the **Scene View**.
- **Text** - links are represented by text boxes with link descriptions in the **Scene View**.

Quick Properties Page

Use the options on this page to customize the way quick properties are displayed in the **Scene View**.

TIP If you want to restore the default values, click the **Defaults** button.

Show Quick Properties Toggles the display of quick properties in the Scene View.

Hide Category Clear this check box to include category names in the quick properties tooltips.

If you don't want to see category names in the quick properties tooltips, select this check box.

Definitions Page

Use the options on this page to set up the quick properties categories.

Buttons

Add Element Click  to add quick properties definitions.

Remove Element Click  to delete the selected quick properties definitions.

Grid View Click  to display quick properties definitions in a tabular format.

List View Click  to display quick properties definitions in a list format.

Records View Click  to display quick properties definitions as records.

Previous and Next Element Use  and  to navigate between quick properties definitions.

If you clicked the **Records View** button, this is the only way to move between the records.

Category Specifies the property category that you want to customize.

Property Specifies the property that is displayed in the tooltips for the selected category.

Developer Page

Use the options on this page to adjust the display of object properties.

Show Internal Properties Indicates whether additional object properties are displayed in Autodesk Navisworks.

Select this check box if you want to get access to the **Geometry** tab and the **Transform** tab in the **Properties** control bar.

Display Page

Use the options on this page to adjust the display performance.

2D Graphics

Level of Detail You can adjust the level of detail of your 2D graphics, which means you can trade off between the rendering performance and 2D fidelity. Select from the following options:

- **Low** - gives you lower 2D fidelity, but better rendering performance.
- **Medium** - gives you medium 2D fidelity, and medium rendering performance; this is the default option.
- **High** - gives you higher 2D fidelity, but lower rendering performance.

Detail

Guarantee Frame Rate Indicates whether the Autodesk Navisworks engine maintains the frame rate specified on the [Speed tab](#) (page 757) of the **File Options** dialog box.

By default, this check box is selected, and the target rate is maintained while moving. When movement stops the complete model is rendered.

If this check box is clear, the complete model is always rendered during navigation, no matter how long it takes.

Fill In Detail Indicates whether Autodesk Navisworks fills in any discarded detail when navigation has stopped.

Graphics System

Auto-Select Autodesk Navisworks supports two graphics systems: Presenter Graphics and Autodesk Graphics. By default, this check box is selected and Autodesk Navisworks controls which graphics system to use. Clear this check box if you want to select the system yourself. This enables the **System** drop-down box.

Hardware Acceleration Select this check box to utilize any available OpenGL hardware acceleration on your video card.

If your video card drivers do not function well with Autodesk Navisworks, clear this check box.

NOTE If your video card does not support OpenGL hardware acceleration, this check box is not available.

System This drop-down box is available when you clear the **Auto-Select** check box. Select from the following options:

- **Presenter** - supports the display of Presenter materials and uses Hardware or Software OpenGL.
- **Autodesk** - supports the display of Autodesk materials and uses Direct3D or Hardware OpenGL.

NOTE 3D models can use either graphics system, Presenter system is the default option. 2D sheets can only use Autodesk Graphics, and will not render without a Direct 3D/OpenGL supported graphics card.

Occlusion Culling Select this check box to enable occlusion culling. This means that Autodesk Navisworks only draws visible objects and ignores any objects located behind other objects.

Selecting this check box improves the display performance when much of the model is not visible. For example, when you're walking down the corridor of a building.

IMPORTANT Occlusion culling can only be used on a machine with an OpenGL 1.5 compliant graphics card. Also, occlusion culling is not used in a 2D workspace.

Heads Up

XYZ Axes Indicates whether the XYZ Axes indicator is displayed in the **Scene View**.

Show Position Indicates whether the Position Readout is displayed in the **Scene View**.

Primitives

Point Size Enter a number from 1 through 9 to set the size (in pixels) of points drawn in the **Scene View**.

Line Size Enter a number from 1 through 9 to set the width (in pixels) of lines drawn in the **Scene View**.

Snap Size Enter a number from 1 through 9 to set the size (in pixels) of snap points drawn in the **Scene View**.

Enable Parametric Primitives Indicates whether Autodesk Navisworks dynamically renders parametric primitives during interactive navigation. Selecting this check box means the level of detail changes during navigation depending on the distance from the camera. Clear this check box to use the default representations of primitives; the level of detail stays the same during navigation.

Transparency

Interactive Transparency Select this check box to render transparent items dynamically during interactive navigation. By default, this check box is clear, therefore, transparent items are only drawn when interaction has stopped.

NOTE If your video card does not support hardware accelerated OpenGL, selecting this check box can affect display performance.

Drivers Page

Use the options on this page to enable/disable available display drivers.

Available Drivers

This is a list of all drivers that Autodesk Navisworks can support. By default, all drivers are selected.

Software (OpenGL) This is a legacy driver, which only works on 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Presenter (OpenGL) This driver supports Presenter graphics system and only works on 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Autodesk (DirectX 9) This driver supports Autodesk graphics system, and works with both 2D and 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Autodesk (DirectX 10) This driver supports Autodesk graphics system, and works with both 2D and 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Autodesk (DirectX 11) This driver supports Autodesk graphics system, and works with both 2D and 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Autodesk (OpenGL) This driver supports Autodesk graphics system, and works with both 2D and 3D geometries. If the check box is clear, Autodesk Navisworks will ignore this driver when rendering geometries.

Autodesk Page

Use the options on this page to adjust effects and materials used in Autodesk Graphics mode.

Autodesk Effects

Shader Style Defines Autodesk shading style on faces. Select from the following options:

- **Basic Material** - realistic display of faces, close to how they would appear in the real world. This is the default option.
- **Gooch** - uses cool and warm colors instead of dark and light to enhance the display of faces that might be shadowed and difficult to see in a realistic display.

Autodesk Materials

Use Fall Back This option gives you control of forcing to use Basic Material instead of Autodesk consistent material. If your graphics card does not function well with Autodesk consistent material, this option will automatically be used.

Use Lod Texture Select this check box if you want to use LOD textures.

Reflection Enabled Select this check box to enable the reflection color for Autodesk consistent material.

Highlight Enabled Select this check box to enable the specular color for Autodesk consistent material.

Bump Enabled Select this option if you want to use a bump map, which makes a rendered object appear to have a bumpy or irregular surface. For example, when you render an object with a bump-mapped material, lighter (whiter) areas of the map appear to be raised and darker (blackier) areas appear to be low. If the image is in color, the gray-scale value of each color is used. Bump mapping increases rendering time significantly but adds to the realism.

Image Library Selects Autodesk consistent material library based on the texture resolution. Choose from the following options:

- **Base Resolution** - basic material library, with resolution of approximately 256 x 256 pixels. This library is installed by default, and is required by Autodesk Navisworks to support a full range of visual style and color style functionality.
- **Low Resolution** - low resolution images, approximately 512 x 512 pixels.
- **Medium Resolution** - medium resolution images, approximately 1024 x 1024 pixels.
- **High Resolution** - high resolution images. This option is not currently supported.

Max Texture Dimensions This option affects the visual details of the textures applied to geometry. Enter the desired value in pixels. For example, a value of '128' means the maximum texture size of 128 pixels x 128 pixels. The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Procedural Texture Size This option gives the size of textures generated from procedural maps. For example, a value of '256' means the texture size of 256 x 256 pixels generated from procedural maps. The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Multi Sample Anti Aliasing

MSAA Level Defines the value of anti-aliasing to render in Autodesk Graphics mode. Anti-aliasing is used to smooth the edges of the geometry. The higher the number, the smoother the geometry, but the longer the rendering will take. **2x** is the default option.

NOTE If your video card does not support higher MSAA, use lower MSAA that your video card can support automatically.

3Dconnexion Page

Use the options on this page to customize the behavior of 3Dconnexion devices.

NOTE All options are selected by default. If you make any changes you can click the **Default button** to reset to the original settings.

These options are offered in addition to the adjustments that can be made using the **Control Panel** for the device which is supplied by the device manufacturer with the installation.

Speed Use the slider to adjust the sensitivity of the controller.

Keep Scene Upright Select this check box to disable the rolling axis. When selected you will not be able to roll the model sideways.

Center Pivot on Selection Select this check box to move the pivot point to the center of any selections you make.

Pan/Zoom Select this check box to turn on pan and zoom functionality for the 3Dconnexion device.

Tilt/Spin/Roll Select this check box to turn on tilt, spin and roll functionality for the 3Dconnexion device.

Navigation Bar Page

Use the options on this page to customize the behavior of tools on the navigation bar.

Orbit Tools

Use Classic Orbit Select this check box if you want to switch from the standard Orbit tool to the classic Autodesk Navisworks Orbit mode on the navigation bar.

Use Classic Free Orbit (Examine) Select this check box if you want to switch from the standard Free Orbit tool to the classic Autodesk Navisworks Examine mode on the navigation bar.

Use Classic Constrained Orbit (Turntable) Select this check box if you want to switch from the standard Constrained Orbit tool to the classic Autodesk Navisworks Turntable mode on the navigation bar.

Walk Tool

Use Classic Walk Select this check box if you want to switch from the standard Walk tool to the classic Autodesk Navisworks Walk mode on the navigation bar.

Constrain Walk Angle When this check box is selected, the Walk tool will keep the camera upright while navigating. If this check box is clear, the tool

will allow the camera to roll while navigating (resulting in behavior almost like the Fly tool).

Use Viewpoint Linear Speed When this check box is selected, the Walk tool will respect the Viewpoint Linear Speed setting. In this case, the Walk speed slider will act like a multiplier.

When this check box is clear, the Walk tool will work independently of the Viewpoint Linear Speed setting, using a fixed value set with the slider.

Walk Speed Sets the speed of the Walk tool from 0.1 (very slow) to 10 (very fast).

ViewCube Page

Use the options on this page to customize the ViewCube behavior.

Show the ViewCube Indicates whether or not the ViewCube is displayed in the **Scene View**.

TIP You can also toggle the ViewCube by clicking View tab ► Navigation Aids panel ► ViewCube .

Size Specifies the size of the ViewCube. You can choose from the following options:

- Automatic
- Tiny
- Small
- Medium
- Large

NOTE In automatic mode, the size of the ViewCube is relative to the size of the **Scene View**, and ranges between medium and tiny.

Inactive Opacity When the ViewCube is inactive, that is your cursor is distant from the ViewCube, it appears transparent. To control the opacity level, choose from the following options:

- 0%
- 25%
- 50%
- 75%

■ 100%

Keep Scene Upright Indicates whether the upside-down orientations of the scene is allowed when you use the ViewCube.

When this check box is selected, dragging the ViewCube produces a turntable effect.

When Dragging on the ViewCube

While being dragged, the ViewCube and the scene rotate in an arcball like fashion, unless the **Keep Scene Upright** check box is selected.

Snap to the Closest View Indicates whether the ViewCube snaps to one of the fixed views when it is angularly close to one of the fixed views.

When Clicking on the ViewCube

Fit-to-View on Change When this check box is selected, clicking the ViewCube rotates around the center of the scene and zooms out to fit the scene into the Scene View. When dragging the ViewCube, prior to the drag, the view changes to look at the scene center (but does not zoom) and continues to use that as the pivot point while dragging.

If this check box is clear, clicking or dragging the ViewCube rotates around the current pivot point and does not zoom in or out.

Use Animated Transitions When Switching Views If this check box is selected, an animated transition displays when you click on a section of the ViewCube to help you visualize the spatial relationship between the current viewpoint and the selected viewpoint.

NOTE When navigating about 3D scenes that contain vast amounts of geometry, the application frame rate may drop and make it difficult for the system to smoothly animate a viewpoint transition.

Show the Compass Below the ViewCube Indicates whether the compass is displayed below the ViewCube tool.

SteeringWheels

Use the options on this page to customize the SteeringWheels menus.

Big Wheels

Size Specifies the size of big wheels. You can choose from the following options:

- Small (64x64)
- Normal (128x128)
- Large (256x256).

Normal is the default option.

Opacity Controls the opacity level of big wheels. The default value is 50%. You can choose from the following options:

- 25% (mostly transparent)
- 50%
- 75%
- 90% (mostly opaque)

Mini Wheels

Size Specifies the size of mini wheels. You can choose from the following options:

- Small (16x16)
- Normal (32x32)
- Large (64x64).
- Extra Large (256x256)

Normal is the default option.

Opacity Controls the opacity level of mini wheels. The default value is 50%. You can choose from the following options:

- 25% (mostly transparent)
- 50%
- 75%
- 90% (mostly opaque)

On-Screen Messages

Show Tool Messages Toggles the display of tooltips for navigation tools. When this check box is selected, the tooltips are shown below the cursor as you use the tools.

NOTE This setting is always on for View Object and Tour Building wheels, and cannot be turned off.

Show Tooltips Toggles the display of wheel tooltips. When this check box is selected, the tooltips are shown when you hover over wedges on the wheels.

NOTE This setting is always on for View Object and Tour Building wheels, and cannot be turned off.

Show Tool Cursor Text Toggles the display of tool label below the cursor.

NOTE This setting is always on for View Object and Tour Building wheels, and cannot be turned off.

Look Tool

Invert Vertical Axis Selecting this check box swaps the up-down axis for the Look tool; that is pushing the mouse forward looks down, and pulling the mouse backward looks up.

Walk Tool

Constrain Walk Angle Selecting this check box makes the Walk Tool respect the world up vector (as set in File Options ► Orientation). As a result, using the Walk tool causes the camera to snap to the current up vector.

When this check box is clear, the Walk tool disregards the world up vector, and the camera is walked with its current up orientation unaffected.

Use Viewpoint Linear Speed When this check box is selected, the Walk tool will respect the Viewpoint Linear Speed setting. In this case, the Walk speed slider will act like a multiplier.

When this check box is clear, the Walk tool will work independently of the Viewpoint Linear Speed setting, using a fixed value set with the slider.

Walk Speed Sets the speed of the Walk tool from 0.1 (very slow) to 10 (very fast).

Zoom Tool

Enable Single-Click Incremental Zoom In When this check box is selected, single clicking over the Zoom wedge increases the magnification of the model.

When this check box is clear, nothing happens when you single click over the Zoom wedge.

Orbit Tool

Keep Scene Upright When this check box is selected, the Orbit tool behaves similarly to the classic Orbit mode, with orbiting constrained along the XY axis and in the Z direction.

When this check box is clear, the Orbit tool behaves similarly to the classic Examine mode, and you can roll the model around the pivot point.

Center Pivot on Selection When this check box is selected, the objects selected before the Orbit tool are used to calculate the pivot point to use for orbiting. The pivot point is calculated based on the center of the extents of the selected objects.

User Interface Page

Use the options on this page to choose the user interface (standard or classic), and select the color theme.

User Interface Choose between the following options:

- **Classic.** Switches over to the classic Autodesk Navisworks interface with old-style menu and toolbars.
- **Standard (Recommended).** Switches over to the new interface with a ribbon tool palette. This is the default option.

Theme Use the drop-down list to apply one of the preset interface themes.

Model Node

Use the settings in this node to optimize Autodesk Navisworks performance, and customize parameters for NWD and NWC files.

TIP If you want to restore the default values, click the **Defaults** button.

Performance Page

Use the options on this page to optimize Autodesk Navisworks performance.

Memory Limit

Auto Indicates whether Autodesk Navisworks automatically determines the maximum memory that can be used. Selecting this check box sets the memory limit to the lowest of your available physical memory or address space, less that required for your operating system.

Limit (MB) Specifies the maximum memory that Autodesk Navisworks can use.

Merge Duplicates

These options improve performance by multiply instancing matching items. Rather than storing every item in memory, if any items are the same, Autodesk Navisworks can store one instance of them and 'copy' that instance into other positions. This is of particular benefit on larger models, where there are significant numbers of these duplicate geometries.

On Convert Select this check box to merge duplicates when a CAD file is converted into the Autodesk Navisworks format.

On Append Select this check box to merge duplicates when a new file is appended to the currently opened Autodesk Navisworks file.

On Load Select this check box to merge duplicates when a file is loaded into Autodesk Navisworks.

On Save NWF Select this check box to merge duplicates when the current scene is saved in the NWF file format.

On Load

Collapse on Convert Collapses the tree structure on the **Selection Tree** to the specified level when native CAD files are converted into Autodesk Navisworks. Select from the following options:

- **None** - the tree is fully expanded. Use this option to enable splitting polylines into individual segments when importing DWGs and DGNs to support multiple clash intersections. For DGN files, you also need to select **File Readers > DGN > Split Lines** check box, and deselect **File Readers > DGN > Merge Lines and Arcs** check box. For DWG files, you also need to set **File Readers > DWG/DXF > Line Processing** drop-down to **Separate All Lines**.
- **Composite Objects** - the tree is collapsed up to the level of composite objects.
- **All Objects** - the tree is collapsed up to the level of objects.

- **Layers** - the tree is collapsed up to the level of layers.
- **Files** - the tree is collapsed up to the level of files.

This enables performance to be prioritized over structure/properties and has the added benefit of improving streaming by cutting down the logical structure.

NOTE Although Autodesk Navisworks tries to collapse items to the fewest number possible, it may be necessary to prevent collapsing in some cases to preserve model fidelity. For example, if an item has properties or materials unique to itself, then collapsing would endanger this information, and therefore it will not be collapsed.

Close NWC/NWD files on Load Indicates whether NWC and NWD files are closed once they've been loaded into memory.

When you open NWC/NWD files, Autodesk Navisworks locks them for editing. By selecting this check box, you instruct Autodesk Navisworks to close NWC or NWD files as soon as they've been loaded into memory. This means that the files can be opened and edited by other users while you are viewing them.

Create Parametric Primitives Select this check box to enable creation of parametric models (models described by formulae not vertices).

Using this option allows you to get better looking visuals, faster rendering, smaller memory footprint (especially, when loading DGN and RVM files with significant amounts of parametric data that no longer need to be converted into vertices in Autodesk Navisworks).

NOTE Modifying this option takes effect when you next load or refresh file.

Create Presenter Materials Select this check box to enable creation of **Presenter** materials when NWC files are loaded.

Clearing this check box turns off creation of **Presenter** materials.

Temporary File Location

Auto Indicates whether Autodesk Navisworks automatically selects your user Temp folder.

Location Click  to open the **Browse for Folder** dialog box, and select the desired Temp folder.

NWD Page

Use the options on this page to enable and disable geometry compression and select whether the precision of certain options is reduced when saving or publishing NWD files.

Geometry Compression

Enable Select this check box to enable geometry compression when NWD files are saved.

Geometry compression results in less memory being required and therefore smaller NWD files.

Reduce Precision

Coordinates Select this check box to reduce the precision of coordinates.

Precision Specifies the precision value for coordinates. The larger the value, the less precise coordinates are.

Normals Select this check box to reduce the precision of normals.

Colors Select this check box to reduce the precision of colors.

Texture Coordinates Select this check box to reduce the precision of texture coordinates.

See also:

[Native File Format](#) (page 163)

NWC Page

Use the options on this page to manage reading and writing of cache files (NWC).

By default, when Autodesk Navisworks opens a native CAD file (for example, AutoCAD or MicroStation), it first checks in the same directory whether there is a cache file present with the same name as the CAD file but with an .nwc extension. If there is, and this cache file is newer than the native CAD file, then Autodesk Navisworks opens this file instead as it has already been converted to Autodesk Navisworks format and, therefore, opens much quicker. If, however, there is no cache file present, or the cache file is older than the native CAD file, then Autodesk Navisworks has to open the CAD file and

convert it. By default, it writes a cache file in the same directory and with the same name as the CAD file, but with the .nwc extension, for speeding up the opening of this file in future.

Caching

Read Cache Select this check box to use cache files when Autodesk Navisworks opens native CAD files.

Clear this check box if you don't want to use cache files. This ensures that Autodesk Navisworks converts native CAD files each time they are opened.

Write Cache Select this check box to save cache files when native CAD files are converted. Generally, cache files are much smaller than original CAD files, therefore, selecting this option does not take up too much disk space.

Clear this check box if you don't want to save cache files.

Geometry Compression

Enable Select this check box to enable geometry compression when NWC files are saved.

Geometry compression results in less memory being required and therefore smaller NWC files.

Reduce Precision

Coordinates Select this check box to reduce the precision of coordinates.

Precision Specifies the precision value for coordinates. The larger the value, the less precise coordinates are.

Normals Select this check box to reduce the precision of normals.

Colors Select this check box to reduce the precision of colors.

Texture Coordinates Select this check box to reduce the precision of texture coordinates.

See also:

[Native File Format](#) (page 163)

File Readers Node

Use the settings in this node to configure the file readers required to open native CAD and scanning applications file formats in Autodesk Navisworks.

TIP If you want to restore the default values, click the **Defaults** button.

3DS Page

Use this page to adjust the options for the 3DS file reader.

Convert Hidden Select this check box if you want Autodesk Navisworks to convert hidden entities in 3DS files. They are automatically marked as hidden by Autodesk Navisworks.

When this check box is clear, the file reader ignores hidden items.

Bitmap File Search Paths The paths of texture map files are not stored with the texture maps in the model. Enter the required paths into this box, separating them with semi-columns.

Default Units Specifies the type of units Autodesk Navisworks uses when opening 3DS files.

TIP If the chosen units type turns out to be wrong, you can rescale the model by using the **Units and Transform** option. (Right-click in the **Scene View**, and click **Units and Transform** on the shortcut menu.)

See also:

[3DS File Reader](#) (page 170)

ASCII Laser Page

Use this page to adjust the options for the ASCII laser scan file reader.

Sample Rate Specifies the frequency of points extracted from the input file. Increasing the sample rate, reduces the number of points extracted. This has the effect of reducing the image resolution, but increasing the speed with which the file is loaded.

Use Point Intensity Values Select the check box to extract intensity values from the input file.

When this check box is clear, the file reader ignores intensity values.

Use Point Color Values Select the check box to extract color values from the input file.

When this check box is clear, the file reader ignores color values.

See also:

[ASCII Laser Scan File Reader](#) (page 171)

CIS/2 Page

Use this page to adjust the options for the CIS/2 file reader.

Convert Features Indicates whether features are converted.

NOTE Converting a large number of features may have a negative impact on the loading performance.

Convert Unknown Section into Rectangle Select this check box to convert unknown sections profiles into rectangles with estimated parameters, but the scale may not be appropriate.

When this check box is clear, the file reader ignores unknown sections profiles.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry. If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

DGN Page

Use this page to adjust the options for the 3D DGN and PROP file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Convert Hidden Items Select this check box to convert hidden entities in DGN files. They are automatically marked as hidden by Autodesk Navisworks. When this check box is clear, the file reader ignores hidden items.

Show Hidden Items Select this check box, if you want to display all entities in converted DGN files, irrespective of whether they are hidden or not.

NOTE To use this option, you must select the **Convert Hidden Items** check box.

Convert Lines and Arcs Select this check box to convert lines, splines, curves, arcs, circles, or ellipses in DGN files.

Merge Lines and Arcs Select this check box if you want to reduce the complexity of the model in the **Selection Tree** by interpreting adjoining lines with the same color, level, and parent as a single item.

Clear this check box if you want to leave these elements as separate items in Autodesk Navisworks.

Split Lines This option instructs the file reader to ungroup polyline objects. As a result, line elements are split into separate nodes for each segment of the line.

Use this option when you need to enhance clash detection analysis. By default, **Clash Detect** treats multi-segment entities as a single object, reporting one clash for each object pair. Ungrouping polyline objects means that each line segment can be clashed independently from other segments of the line. As a result, all potential clashes are reported, and not just the first one found.

IMPORTANT In order for this function to work properly, you must clear the **Merge Lines and Arcs** check box, as it takes priority over the **Split Lines** option.

Additionally, you need to set the **Model > Performance > Collapse on Convert** drop-down to **None**, otherwise the polyline objects will be collapsed into a single geometry node.

Convert Text Select this check box to convert text into quick properties in Autodesk Navisworks.

When this check box is clear, the file reader ignores text in DGN files.

Default Font Indicates which font to use for the converted text.

Shape Merge Threshold Specifies the number of vertices for MicroStation shapes.

If the number of vertices is less than or equal to the **Shape Merge Threshold** value, Autodesk Navisworks merges all shapes on the same level or in the same cell and with the same color into a “Shape Set”.

Convert References Select this check box to convert reference files in DGN files.

When this check box is clear, the file reader ignores reference files.

Ignore Unres. References Select this check box to ignore unresolved reference files in DGN files.

If this check box is clear, the file reader attempts to convert all reference files, and presents you with a dialog box to find any unresolved reference files at a run time.

Use Level Symbology Select this check box to enable the level symbology from MicroStation. This makes items in Autodesk Navisworks take their color from level rather than use the default element color in MicroStation.

Align Global Origins Indicates whether Autodesk Navisworks aligns global origins in referenced DGN files to the origin of the master DGN file.

When DGN files are appended together with this check box selected, their global origins are positioned in the same place.

Use Materials Indicates whether MicroStation's materials are exported and assigned to elements.

Select this check box to assign the same textures, diffuse, ambient, and specular colors to the elements as in the MicroStation scene.

Clear this check box to assign the same colors to the elements as in the MicroStation scene.

Material Search Paths Enter a semi-colon separated list of paths to the MicroStation palette (.pal) and material (.mat) files. These paths are used by the file reader to convert MicroStation materials.

Convert PDS Data Select this check box to read object information from Intergraph's Plant Design System™ while converting the DGN files.

PDS information is read from Intergraph's DRV files. The file reader looks for a DRV file with the same base name as the DGN file in the same directory.

When this check box is clear, the file reader ignores PDS data.

Convert TriCAD Data Select this check box to read object information from Triplan's TriCAD™ while converting DGN files.

When this check box is clear, the file reader ignores TriCAD data.

Convert TriForma Data Select this check box to read object information from Bentley's TriForma™ while converting DGN files.

When this check box is clear, the file reader ignores TriForma data.

TriForma Dataset Search Paths Enter a semi-colon separated list of paths that the file reader uses to locate TriForma data set files. You must include the paths to user-defined datasets.

View Number Enter the required value to select a specific view for loading. The file reader uses the level visibility of this view when converting items.

For example, enter 0 to use the first active view.

See also:

[DGN File Reader](#) (page 180)

DWF Page

Use this page to adjust the options for the DWF file reader.

Faceting Factor in 3D Models Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double the value. To get half as many facets, halve the value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation in 3D Models (m) This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation in 3D Models (m)** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation in 3D Models (m)** is set to 0, then this function is ignored.

Extract Textures in 3D Models Select this check box to extract textures and environment maps associated with the file.

NOTE Environment maps are not set in the scene by default, and must be configured in **Presenter** manually.

When this check box is clear, the file reader ignores maps and textures.

See also:

[DWF File Reader](#) (page 179)

DWG/DXF Page

Use this page to adjust the options for the DWG/ DXF file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double the value. To get half as many facets, halve the value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Split by Color Compound objects can be split into parts based on their color. Select this check box if you want to use this functionality.

For example, a window object from Architectural Desktop can be split into a frame and a pane. If this check box is clear, you can only select the window object as a whole, whereas if you select this check box, you can select the individual pane and frame.

NOTE Autodesk Navisworks names parts of compound objects after their color.

Default Decimal Units Selects the types of units Autodesk Navisworks uses to open DWG and DXF files that were created with decimal drawing units.

NOTE DWG and DXF files do not specify the units they were created in. To adjust the units in Autodesk Navisworks, use the **Units and Transform** option.

Merge 3D Faces Indicates whether the file reader interprets adjoining faces with the same color, layer, and parent as a single item in the **Selection Tree**. Clear this check box to keep the entities as separate items in the **Selection Tree**.

Line Processing Specifies how the file reader processes lines and polylines. Select one of the following options:

- **Merge Lines by Color.** This option merges any lines on the same layer or on the same proxy entity that match by color. Use this option when you want more efficient file processing and navigation.
- **As Provided.** This option reads lines and polylines the same way as they are specified by the original DWG.
- **Separate All Lines.** This option splits line elements into separate nodes for each segment of the line.

Use this option when you need to enhance clash detection analysis. By default, **Clash Detective** treats multi-segment entities as a single object, reporting one clash for each object pair. Ungrouping polyline objects means that each line segment can be clashed independently from other segments of the line. As a result, all potential clashes are reported, and not just the first one found.

IMPORTANT In order for this function to work properly, you need to set the **Model > Performance > Collapse on Convert** drop-down to **None**, otherwise the polyline objects will be collapsed into a single geometry node.

Convert Off Select this check box to convert layers that are switched off in DWG and DXF files. They are automatically marked as hidden in Autodesk Navisworks.

When this check box is clear, the file reader ignores switched off layers.

Convert Frozen Select this check box to convert layers that are frozen in DWG and DXF files. They are automatically marked as hidden in Autodesk Navisworks.

When this check box is clear, the file reader ignores frozen layers.

Convert Entity Handles Select this check box to convert entity handles, and attach them to object properties in Autodesk Navisworks.

When this check box is clear, the file reader ignores entity handles.

Convert Groups Select this check box to retain groups in DWG and DXF files; this adds another selection level to the **Selection Tree**.

When this check box is clear, the file reader ignores groups.

Convert XRefs Select this check box to convert any external reference files contained within the DWG file automatically.

Clear this check box if you want to append files in Autodesk Navisworks later yourself.

Merge XRef Layers Select this check box to merge the layers in external reference files with layers in the main DWG file in the **Selection Tree**.

Clear this check box to keep the external reference files separate from the main DWG file in the **Selection Tree**.

Convert Views Select this check box to convert the named views into Autodesk Navisworks viewpoints.

When this check box is clear, the file reader ignores views.

Convert Points Select this check box to convert points in DWG and DXF files.

When this check box is clear, the file reader ignores points.

Convert Lines Select this check box to convert lines and arcs in DWG and DXF files.

When this check box is clear, the file reader ignores lines.

Convert Snap Points Select this check box to convert snap points in DWG and DXF files.

When this check box is clear, the file reader ignores snap points.

Convert Text Select this check box to convert text in DWG and DXF files.

When this check box is clear, the file reader ignores text.

Default Font Indicates which font to use for the converted text.

Convert Point Clouds Select this check box to convert the AutoCAD Point Cloud entities. This applies to AutoCAD and Autodesk Navisworks 2011 or later. Earlier versions will ignore the point clouds or display proxy graphics (often as a wireframe box).

When this check box is clear, the file reader ignores point clouds.

Point Cloud Detail Specifies how much detail to extract from the point cloud. The valid entries are between 1 and 100, where 100 = all the points, 10 = about 10% of the points, 1 = about 1% of the points.

Use Point Cloud Colors Controls the point cloud colors. Select this check box to use color values for the points in the point cloud.

When this check box is clear, any color values for the points in the point cloud are ignored and the normal AutoCAD color for the entity is used. This option is useful if the specific colors stored are too dark or meaningless.

DWG Loader Version Specifies which version of ObjectDBX to use when loading AutoCAD files. This option enables you to select support for the correct version of object enablers that can be used within the file.

NOTE When you modify this setting, restart Autodesk Navisworks to apply the changes. This is because once any particular version of ObjectDBX is loaded during reading of a DWG or DXF file, the version in use does not change until the program is restarted.

Load Material Definitions Select this check box to extract material definitions from DWG files.

When this check box is clear, the file reader ignores material definitions.

Use ADT Standard Configuration Select this check box to convert geometry and materials in DWG files using the Standard display configuration.

Clear this check box to convert geometry and materials accordingly to whether they are displayed in the currently saved display configuration.

Convert Hidden ADT Spaces Indicates whether space objects that lack any visible 3D geometry in DWG files are converted (for example, objects that lack floor or ceiling thicknesses).

Selecting this check box results in corresponding hidden objects appearing in Autodesk Navisworks.

NOTE This option does not affect the normal behavior of space objects that have visible 3D geometry in DWG files.

Material Search Paths Autodesk Navisworks automatically searches the default Autodesk material paths.

Use this box to specify additional paths to texture files used in Autodesk Architectural Desktop materials. Separate the paths with semi-columns.

Render Type Specifies the render style used for objects when loading DWG files.

Selecting **Automatic** means Autodesk Navisworks uses the render style saved in DWG files.

If geometry is not correctly displayed, adjust the render style with one of the following options: **Rendered**, **Shaded**, or **Wireframe**.

Buttons

Defaults Restores the default values.

Advanced Opens the [Convert Object Properties Dialog Box](#) (page 743) where you can select third-party applications to read object information from.

See also:

[DWG/DXF File Reader](#) (page 175)

Faro Page

Use this page to adjust the options for the Faro Scan file reader.

Point Colors Specifies how points are extracted from the input file.

Select from the following options:

- **None** - the points come through as white.
- **Intensity** - the points use the intensity values stored in the file.
- **Color** - the points use the color values stored in the file.

NOTE If the selected option is not available, then the file reader defaults to the values used in the file. For example, if you set **Point Colors** to **Color**, but only intensities are available in the file, then the file reader uses the intensities.

See also:

[Faro Scan File Reader](#) (page 181)

FBX Page

Use this page to adjust the options for the FBX file reader.

Convert Autodesk Materials FBX files can contain either Autodesk or Native materials. Select this check box to convert Autodesk materials. The exporter will attempt to convert Autodesk materials, if they are available. Clear this check box if you want to convert Native FBX materials to Presenter materials.

NOTE Presenter materials are only available when using Presenter graphics system, and Autodesk materials (also referred to as consistent materials) are only available when using Autodesk graphics system. See [Graphics System](#) (page 155).

Convert Lights Select this check box to convert lights.

When this check box is clear, the file reader ignores lights.

Convert Skeletons Select this check box to convert skeletons.

When this check box is clear, the file reader ignores skeletons.

TIP A common technique for animating a 3D model consists of creating a hierarchical articulated structure, named skeleton, whose deformation derives the deformation of associated model. The location and displacement of the skeleton's joints dictate how the model moves.

Convert Textures Select this check box to convert textures.

When this check box is clear, the file reader ignores textures.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

See also:

[FBX File Reader](#) (page 181)

IFC Page

Use this page to adjust the options for the IFC file reader.

Convert Bounding Boxes Select this check box to bring through and visualize bounding boxes.

When this check box is clear, the file reader ignores bounding boxes.

Convert Spaces Select this check box to bring through and visualize spaces.

When this check box is clear, the file reader ignores spaces.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. When you double the value, you get twice the number of facets. When you halve the value you get half as

many facets. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Representation Detail Specifies the level of visual representation of IFC elements.

IFC elements can have multiple visual representations, such as bounding boxes (simplest), lines, styled lines, polygons, and styled polygons (most complex). Loading and showing all of these representations can lead to cluttered visuals and memory overheads.

Select from the following options:

- **Highest Only** - to load and show the most complex level of detail available while ignoring simpler ones.
- **Show Highest** - to load all representations, but only show the highest level of detail available.
- **Show All** - to load and show everything available.

Show Spatial Hierarchy Select this check box to show IFC model as a tree structure in the **Selection Tree**.

Clear this check box to show IFC model as a simple list of elements in the **Selection Tree**.

Use Property-Based Colors Select this check box to convert and use property-based colors.

TIP If an IFC file is predominantly black when loaded, clear this check box to revert back to using IFC-standard colors.

See also:

[IFC File Reader](#) (page 182)

Inventor Page

Use this page to adjust the options for the Inventor file reader.

Active Project Specifies the path of the current Inventor project.

Convert Work Surfaces Select this check box to convert Work Surfaces. When this check box is clear, the file reader ignores Work Surfaces. This is the default option.

Load assembly with last active representation Select this check box to load Inventor assembly with the last active representation. When this check box is clear Inventor files are opened as in Autodesk Navisworks 2011.

See also:

[Inventor File Reader](#) (page 183)

Leica Page

Use this page to adjust the options for the Leica Scan file reader.

Sample Rate Specifies the frequency of points extracted from the input file. Increasing the sample rate, reduces the number of points extracted. As a result, the image resolution is reduced, and the file is converted faster.

Point Colors Specifies how points are extracted from the input file.

Select from the following options:

- **None** - the points come through as white.
- **Raw Intensity** - the points use the intensity values stored in the file.
- **Color** - the points use the color values stored in the file.
- **Color-Mapped Intensity** - the points use a spectrum of RGB colors.
- **Gamma Corrected Intensity** - the points use gamma-corrected raw intensity values stored in the file.

Gamma Correction Level Specifies a gamma correction value used to adjust the normalized point intensity values obtained from the file, and results in lighter or darker images depending on the setting.

A gamma correction value in the range between 0.1 and 0.99 adjusts the weighting of intensity values at the lower end of the intensity range, so that the image appears lighter.

A gamma correction level above 1.0 has the effect of darkening the image.

NOTE A gamma correction of 1 means no changes.

See also:

[Leica Scan File Reader](#) (page 185)

JTOpen Page

Use this page to adjust the options for the JTOpen file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Retriangulate JT Brep Models Select this check box to use BREP entities to create geometry.

Clear this check box to use tri-strips to create geometry.

See also:

[JTOpen File Reader](#) (page 184)

MAN Page

Use this page to adjust the options for the MAN file reader.

Include Line-Geometry Select this check box to convert MicroGDS line-primitives.

When this check box is clear, the file reader ignores line-geometry.

Set Facet Circle Specifies the number of facets for arcs.

Enter the number of straight line segments required to facet a whole circle. This option corresponds to the MicroGDS **Set Facet** preference.

Texture Path Specifies the path to the folder containing images used in MicroGDS Materials. This path is used as the base for relative paths for materials that use image files.

This option corresponds to the MicroGDS **Renderer Textures** preference.

Define Presenter Materials Select this check box to define MicroGDS material-styles as **Presenter** materials in Autodesk Navisworks. When this check box is clear, the file reader ignores MicroGDS material-styles.

See also:

[MAN File Reader](#) (page 185)

Parasolid Page

Use this page to adjust the options for the Parasolid file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry. If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Scale The scale factor to apply to the model. The scale factor must be greater or equal to 0, where 0 results in the scale factor being turned off.

Default Units Specifies the type of units Autodesk Navisworks uses when opening Parasolid files.

TIP If the chosen units type turns out to be wrong, you can rescale the model by using the **Units and Transform** option. (Right-click in the **Scene View**, and click **Units and Transform** on the shortcut menu.)

See also:

[Parasolid File Reader](#) (page 187)

PDS Page

Use this page to adjust the options for the PDS file reader.

Load Tags Select this check box if you want Autodesk Navisworks to read associated TAG files together with the DRI file.

When this check box is clear, the file reader ignores TAG files.

Load Display Sets Select this check box if you want Autodesk Navisworks to read associated Display Set DST files with the DRI file.

When this check box is clear, the file reader ignores Display Set DST files.

Input Files Selects the input file type.

Select from the following options:

■ **DGN Files** - to convert the original DGN files.

■ **NWC Files** - to open the NWC cache versions of the files.

This option is useful if you routinely perform a batch conversion of the DGN files into NWC files, and would like to open the NWC files when reading a DRI file.

See also:

[PDS File Reader](#) (page 187)

Riegl Page

Use this page to adjust the options for the Riegl Scan file reader.

Use Point Color Values Select this check box to extract color values from the input file.

When this check box is clear, the file reader ignores color values.

Use Point Intensity Values Select this check box to extract intensity values from the input file.

When this check box is clear, the file reader ignores intensity values.

Triangulate Point Data Select this check box to extract triangles from the input file.

Selecting this check box slows down the speed of file conversion.

Apply Scanner Transformation Select this check box to display the image in local coordinates relative to the scanner.

Clear this check box to display the image in global coordinates. Use this option when the file contains more than one frame.

See also:

[Riegl Scan File Reader](#) (page 188)

RVM Page

Use this page to adjust the options for the RVM file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. When you double the value, you get twice the number of facets. When you halve the value you get half as many facets. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Convert Attributes Select this check box to convert attribute files. All files located by the file reader are considered in the same order they were found. Attributes are matched to elements in the scene by name. Any attributes that were not found are counted, and the number reported in the scene statistics. When this check box is clear, the file reader ignores attribute files.

Search All Attribute Files Controls the way the file reader locates attribute files. The file extensions are specified in the **Attribute File Extensions** box.

When this check box is clear (the default option), the file reader checks the directory where the RVM file is located for an attribute file with the same name. If it cannot find an attribute file, then it looks in a subdirectory with the same name as the RVM file for all files with the specified file extensions.

When this check box is selected, the file reader first uses the default search method. If it cannot find any attribute files, it returns to the directory where the RVM reader is located, and looks for all files with the specified file extensions.

Attribute File Extensions Specifies the file extensions that the file reader uses to recognize attribute files. The default extensions are *.ATT, *.ATTRIB, and *.TXT. Separate the values in the list with semi-colons.

Generate Texture Coordinates Select this check box to create texture coordinates for each point in the model.

When this check box is clear, the file reader ignores texture coordinates.

Keep Empty Groups Select this check box to convert groups that do not contain any geometry. Use this option when, for example, these groups contain attributes.

When this check box is clear, the file reader ignores empty groups. This is the default option.

Load RVS File Select this check box to read an RVS file at the same time as the corresponding RVM file. This is the default option.

NOTE To work correctly, the RVS file must have the same name, and path as the associated RVM file, but with an .rvs extension.

When this check box is clear, the file reader ignores RVS files.

RVS Transparencies as Materials Select this check box to attach transparent materials to the objects. Use this option, for example, when large objects in the scene are permanently transparent, and you want to convert them as such.

When this check box is clear (the default option), the file reader ignores RVS transparency settings. Use this option, for example, when you want to override transparency settings in the RVS file.

NOTE You can also override object transparency directly in Autodesk Navisworks. (Click **Item Tools** tab ➤ **Appearance** panel ➤ **Transparency** slider.)

Convert Origins Select this check box to convert component origin points into Autodesk Navisworks snap points.

When this check box is clear, origin points are discarded.

Convert Zero-Thickness Solids Select this check box to convert 3D solids with a zero thickness into 2D geometry.

When this check box is clear, 3D solids with a zero thickness are discarded.

See also:

[RVM File Reader](#) (page 188)

SAT Page

Use this page to adjust the options for the SAT file reader.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry. If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Default Units Specifies the type of units Autodesk Navisworks uses when opening SAT files.

TIP If the chosen units type turns out to be wrong, you can rescale the model by using the **Units and Transform** option. (Right-click in the **Scene View**, and click **Units and Transform** on the shortcut menu.)

See also:

[SAT File Reader](#) (page 189)

SKP Page

Use this page to adjust the options for the SketchUp SKP file reader.

Convert Hidden Items Select this check box to convert hidden entities in SKP files. They are automatically marked as hidden in Autodesk Navisworks. When this check box is clear, the file reader ignores hidden items.

Merge Faces Indicates whether the file reader interprets adjoining faces with the same color, layer, and parent as a single item in the **Selection Tree**. Clear this check box to keep the faces as separate items in the **Selection Tree**.

Merge Lines Indicates whether the file reader interprets joining lines with the same color, layer, and parent as a single item in the **Selection Tree**. Clear this check box to keep the lines as separate items in the **Selection Tree**.

See also:

[SketchUp SKP File Reader](#) (page 189)

STL Page

Use this page to adjust the options for the STL file reader.

Default Units Specifies the type of units Autodesk Navisworks uses when opening STL files.

TIP If the chosen units type turns out to be wrong, you can rescale the model by using the **Units and Transform** option. (Right-click in the **Scene View**, and click **Units and Transform** on the shortcut menu.)

Override Normals Normals control how objects appears when lit. By default, this check box is clear, and the file reader uses the data provided in the STL file, giving the most accurate visible results.

Select this check box if you want Autodesk Navisworks to ignore the normals, and generate the lighting data instead, for example when the supplied model contains invalid lighting data.

NOTE Using this option is less accurate than using real lighting data in STL files.

See also:

[STL File Reader](#) (page 190)

VRML Page

Use this page to adjust the options for the VRML file reader.

Default Units Specifies the type of units Autodesk Navisworks uses when opening VRML world files.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. When you double the value, you get twice the number of facets. When you halve the value you get half as many facets. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Override Normals Normals control how objects appears when lit. By default, this check box is clear, and the file reader uses the data provided in the VRML file, giving the most accurate visible results.

Select this check box if you want Autodesk Navisworks to ignore the normals, and generate the lighting data instead, for example when the supplied model contains invalid lighting data.

NOTE Using this option is less accurate than using real lighting data in VRML files.

Override Orientation By default, this check box is clear, and the file reader processes the geometry information in order indicated in VRML files, which can be clockwise or counter-clockwise.

When this check box is selected, the file reader ignores the order in VRML files, and processes the geometry information counter-clockwise. This option is useful, if your model, when viewed in Autodesk Navisworks, has holes or black areas in it.

Override Switch Statements By default, this check box is clear. This means, the file reader uses the default behavior of the switch statements in VRML files.

For switch statements that are used in VRML scripting, related geometry is only converted if the corresponding switch is turned on. As a result, you can get a mostly empty model in Autodesk Navisworks.

When this check box is selected, the file reader processes all switch statements, and brings through extra geometry.

NOTE Overriding Switch Statements can result, for example, in creating duplicate geometry if the switch statement contains several ways to represent the same object.

See also:

[VRML File Reader](#) (page 191)

Z+F Page

Use this page to adjust the options for the Z+F Scan file reader.

Sample Rate Specifies the frequency of points extracted from the input file.

Increasing the sample rate, reduces the number of points extracted. As a result, the image resolution is reduced, and the file is converted faster.

Remove Spurious Points Select this check box to ignore spurious points in the input file.

When this check box is clear, the file reader extracts spurious points.

Point Intensity Specifies how points are extracted from the input file.

Select from the following options:

- **None** - the points come through as white.
- **Raw Intensity** - the points use the intensity values stored in the file.
- **Color-Mapped Intensity** - the points use a spectrum of RGB colors.
- **Gamma Corrected Intensity** - the points use gamma-corrected raw intensity values stored in the file.

Gamma Correction Level Specifies a gamma correction value used to adjust the normalized point intensity values obtained from the file, and results in lighter or darker images depending on the setting.

A gamma correction value in the range between 0.1 and 0.99 adjusts the weighting of intensity values at the lower end of the intensity range, so that the image appears lighter.

A gamma correction level above 1.0 has the effect of darkening the image.

NOTE A gamma correction of 1 means no changes.

See also:

[Z+F Scan File Reader](#) (page 192)

File Exporters Node

Use the settings in this node to configure the file exporters required to export native Autodesk Navisworks files directly from CAD applications.

TIP If you want to restore the default values, click the **Defaults** button.

DWG Page

Use this page to adjust the options for the DWG file exporter.

Convert Entity Handles Select this check box to convert entity handles, and attach them to object properties in Autodesk Navisworks.

When this check box is clear, the file exporter ignores entity handles.

Convert Frozen Select this check box to convert layers that are frozen in DWG and DXF files. They are automatically marked as hidden in Autodesk Navisworks.

When this check box is clear, the file exporter ignores frozen layers.

Convert Groups Select this check box to retain groups in DWG and DXF files; this adds another selection level to the **Selection Tree**.

When this check box is clear, the file exporter ignores groups.

Convert Hidden ADT Spaces Indicates whether space objects that lack any visible 3D geometry in DWG files are converted (for example, objects that lack floor or ceiling thicknesses).

Selecting this check box results in corresponding hidden objects appearing in Autodesk Navisworks.

NOTE This option does not affect the normal behavior of space objects that have visible 3D geometry in DWG files.

Convert Lines Select this check box to convert lines and arcs in DWG and DXF files.

When this check box is clear, the file exporter ignores lines.

Convert Off Select this check box to convert layers that are switched off in DWG and DXF files. They are automatically marked as hidden in Autodesk Navisworks.

When this check box is clear, the file exporter ignores switched off layers.

Convert Points Select this check box to convert points in DWG and DXF files.

When this check box is clear, the file exporter ignores points.

Convert Snap Points Select this check box to convert snap points in DWG and DXF files.

When this check box is clear, the file exporter ignores snap points.

Convert Text Select this check box to convert text in DWG and DXF files.

When this check box is clear, the file exporter ignores text.

Convert Views Select this check box to convert the named views into Autodesk Navisworks viewpoints.

When this check box is clear, the file exporter ignores views.

Convert XRefs Select this check box to convert any external reference files contained within the DWG file automatically.

Clear this check box if you want to append files in Autodesk Navisworks later yourself.

Default Decimal Units Selects the types of units Autodesk Navisworks uses to open DWG and DXF files that were created with decimal drawing units.

NOTE DWG and DXF files do not specify the units they were created in. To adjust the units in Autodesk Navisworks, use the **Units and Transform** option.

Default Font Specifies default font used to display converted text in Autodesk Navisworks.

Convert Point Clouds Select this check box to convert the AutoCAD Point Cloud entities. This applies to AutoCAD and Autodesk Navisworks 2011 or later. Earlier versions will ignore the point clouds or display proxy graphics (often as a wireframe box).

When this check box is clear, the file reader ignores point clouds.

Point Cloud Detail Specifies how much detail to extract from the point cloud. The valid entries are between 1 and 100, where 100 = all the points, 10 = about 10% of the points, 1 = about 1% of the points.

Use Point Cloud Colors Controls the point cloud colors. Select this check box to use color values for the points in the point cloud.

When this check box is clear, any color values for the points in the point cloud are ignored and the normal AutoCAD color for the entity is used. This option is useful if the specific colors stored are too dark or meaningless.

DWG Loader Version Specifies which version of ObjectDBX to use when loading AutoCAD files.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double the value. To get half as many facets, halve the value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Load Material Definitions Select this check box to extract material definitions from DWG files.

When this check box is clear, the file exporter ignores material definitions.

Material Search Paths Autodesk Navisworks automatically searches the default Autodesk material paths.

Use this box to specify additional paths to texture files used in Autodesk Architectural Desktop materials. Separate the paths with semi-columns.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Merge 3D Faces Indicates whether the file exporter interprets adjoining faces with the same color, layer, and parent as a single item in the **Selection Tree**.

Clear this check box to keep the entities as separate items in the **Selection Tree**.

Line Processing Specifies how the file reader processes lines and polylines. Select one of the following options:

- **Merge Lines by Color.** This option merges any lines on the same layer or on the same proxy entity that match by color. Use this option when you want more efficient file processing and navigation.
- **As Provided.** This option reads lines and polylines the same way as they are specified by the original DWG.
- **Separate All Lines.** This option splits line elements into separate nodes for each segment of the line.

Use this option when you need to enhance clash detection analysis. By default, **Clash Detective** treats multi-segment entities as a single object, reporting one clash for each object pair. Ungrouping polyline objects means that each line segment can be clashed independently from other segments of the line. As a result, all potential clashes are reported, and not just the first one found.

IMPORTANT In order for this function to work properly, you need to set the **Model > Performance > Collapse on Convert** drop-down to **None**, otherwise the polyline objects will be collapsed into a single geometry node.

Merge XRef Layers Select this check box to merge the layers in external reference files with layers in the main DWG file in the **Selection Tree**.

Clear this check box to keep the external reference files separate from the main DWG file in the **Selection Tree**.

Render Type Specifies the render style used for objects when loading DWG files.

Selecting **Automatic** means Autodesk Navisworks uses the render style saved in DWG files.

If geometry is not correctly displayed, adjust the render style with one of the following options: **Rendered**, **Shaded**, or **Wireframe**.

Split by Color Compound objects can be split into parts based on their color. Select this check box if you want to use this functionality.

For example, a window object from Architectural Desktop can be split into a frame and a pane. If this check box is clear, you can only select the window object as a whole, whereas if you select this check box, you can select the individual pane and frame.

NOTE Autodesk Navisworks names parts of compound objects after their color.

Use ADT Standard Configuration Select this check box to convert geometry and materials in DWG files using the Standard display configuration.

Clear this check box to convert geometry and materials accordingly to whether they are displayed in the currently saved display configuration.

See also:

[AutoCAD File Exporter](#) (page 192)

Revit Page

This page enables you to adjust the NWC export options for the Revit file exporter.

Convert Construction Parts When using the Revit 2012 Construction Modelling and Assemblies feature, you have an option of exporting either the original object or construction parts into Autodesk Navisworks. Select this box if you want to export the construction parts; clear this check box if you want to export the original object.

Convert Element Ids Select this check box to export the ID numbers for each Revit element.

When this check box is clear, the file exporter ignores IDs.

Convert Element Parameters Specifies how Revit parameters are read. Select from the following options:

- **None** - the file exporter does not convert parameters.
- **Elements** - the file exporter converts parameters for all found elements.

- **All** - the file exporter converts parameters for all found elements, including the referenced elements. As a result, extra property tabs are available in Autodesk Navisworks.

Convert Linked Files Revit projects can embed external files as links. When this check box is selected, the linked files will be included in the exported NWC file. By default, this check box is clear.

NOTE Only linked Rvt files can be exported; the linked dwg and any other file formats are not supported.

Convert Room as Attribute Indicates whether rooms attributes are supported. By default, this check box is selected, and data for each room converts into a single shared room attribute.

Convert URLs Indicates whether URL property data is converted. By default, this check box is selected and the hyperlinks are supported in the converted file.

Coordinates Specifies whether to use shared or internal coordinates for file aggregation. By default, shared coordinates will be used. Shared coordinates can be viewed and modified outside of Revit. **Export** Specifies how geometry is exported. Select from the following options:

- **Entire Project** - all geometry in the project is exported.
- **Current View** - everything that is currently visible is exported.
- **Selection** - only the currently selected geometry is exported.

Export Room Geometry Indicates whether room geometry is exported.

Try and Find Missing Materials When this check box is selected (the default option), the file exporter looks for a match for the materials missing from the export.

NOTE If any inappropriate materials are applied to the model geometry as a result, clear this check box to fix the problem.

Units for Area/Volume Properties Selects the type of units Autodesk Navisworks uses to convert area and volume-based properties.

See also:

[Revit File Exporter](#) (page 200)

DGN Page

Use this page to adjust the options for the 3D DGN file exporter.

Align Global Origins Indicates whether Autodesk Navisworks aligns global origins in referenced DGN files to the origin of the master DGN file.

When DGN files are appended together with this check box selected, their global origins are positioned in the same place.

Convert Hidden Items Select this check box to convert hidden entities in DGN files. They are automatically marked as hidden by Autodesk Navisworks. When this check box is clear, the file exporter ignores hidden items.

Convert Lines and Arcs Select this check box to convert lines, splines, curves, arcs, circles, or ellipses in DGN files.

Convert PDS Data Select this check box to read object information from Intergraph's Plant Design System™ while converting the DGN files.

PDS information is read from Intergraph's DRV files. The file exporter looks for a DRV file with the same base name as the DGN file in the same directory. When this check box is clear, the file exporter ignores PDS data.

Convert References Select this check box to convert reference files in DGN files.

When this check box is clear, the file exporter ignores reference files.

Convert Text Select this check box to convert text into quick properties in Autodesk Navisworks.

When this check box is clear, the file exporter ignores text in DGN files.

Convert TriCAD Data Select this check box to read object information from Triplan's TriCAD™ while converting DGN files.

When this check box is clear, the file exporter ignores TriCAD data.

Convert TriForma Data Select this check box to read object information from Bentley's TriForma™ while converting DGN files.

When this check box is clear, the file exporter ignores Tri-Forma data.

Default Font Specifies default font used to display converted text in Autodesk Navisworks.

Faceting Factor Enter the required value to control the level of faceting that takes place.

The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1. To get twice the number of facets, double this value. To get half as many facets, halve this value. Larger faceting factors result in more polygons to a model and larger Autodesk Navisworks files.

Ignore Unres. References Select this check box to ignore unresolved reference files in DGN files.

If this check box is clear, the file exporter attempts to convert all reference files, and presents you with a dialog box to find any unresolved reference files at a run time.

Material Search Paths Enter a semi-colon separated list of paths to the MicroStation palette (.pal) and material (.mat) files. These paths are used by the file exporter to convert MicroStation materials.

Max Facet Deviation This setting controls the greatest distance between the edge of a facet and the actual geometry.

If this distance is greater than the **Max Facet Deviation** value, Autodesk Navisworks adds more facets.

If the **Max Facet Deviation** is set to 0, then this function is ignored.

Merge Lines and Arcs Select this check box if you want to reduce the complexity of the model in the **Selection Tree** by interpreting adjoining lines with the same color, level, and parent as a single item.

Clear this check box if you want to leave these elements as separate items in Autodesk Navisworks.

Split Lines This option instructs the file reader to ungroup polyline objects. As a result, line elements are split into separate nodes for each segment of the line.

Use this option when you need to enhance clash detection analysis. By default, **Clash Detect** treats multi-segment entities as a single object, reporting one clash for each object pair. Ungrouping polyline objects means that each line segment can be clashed independently from other segments of the line. As a result, all potential clashes are reported, and not just the first one found.

IMPORTANT In order for this function to work properly, you must clear the **Merge Lines and Arcs** check box, as it takes priority over the **Split Lines** option. Additionally, you need to set the **Model > Performance > Collapse on Convert** drop-down to **None**, otherwise the polyline objects will be collapsed into a single geometry node.

Shape Merge Threshold Specifies the number of vertices for MicroStation shapes.

If the number of vertices is less than or equal to the **Shape Merge Threshold** value, Autodesk Navisworks merges all shapes on the same level or in the same cell and with the same color into a “Shape Set”.

Show Hidden Items Select this check box, if you want to display all entities in converted DGN files, irrespective of whether they are hidden or not.

NOTE To use this option, you must select the **Convert Hidden Items** check box.

TriForma Dataset Search Paths Enter a semi-colon separated list of paths that the file reader uses to locate Triforma data set files. You must include the paths to user-defined datasets.

Use Level Symbolology Select this check box to enable the level symbology from MicroStation. This makes items in Autodesk Navisworks take their color from level rather than use the default element color in MicroStation.

Use Materials Indicates whether MicroStation's materials are exported and assigned to elements.

Select this check box to assign the same textures, diffuse, ambient, and specular colors to the elements as in the MicroStation scene.

Clear this check box to assign the same colors to the elements as in the MicroStation scene.

View Number Enter the required value to select a specific view for loading. The file exporter uses the level visibility of this view when converting items. For example, enter 0 to use the first active view.

See also:

[MicroStation File Exporter](#) (page 201)

Viz/Max Page

Use this page to adjust the options for the 3DS file exporter.

Convert Hidden Items Select this check box to convert hidden entities in 3DS files. They are automatically marked as hidden by Autodesk Navisworks. When this check box is clear, the file exporter ignores hidden items.

Convert User Properties Select this check box to attach any user properties you have defined in Viz or Max to the converted Autodesk Navisworks items. When this check box is clear, the file exporter ignores user properties.

Pre-Render Scene By default, this check box is clear.

Select this check box if the exported file misses some texture maps. This forces Viz/Max to do an internal render, and cache all texture maps. As a result, all texture maps are exported with the model.

See also:

[Viz and Max File Exporter](#) (page 205)

Tools Node

Use the settings in this node to adjust the options for **Clash Detective**, **Presenter**, **TimeLiner**, **Scripter**, **Animator**, and the **Vault** add-in.

Clash Detective Page

Use this page to adjust the **Clash Detective** options.

View in Context Zoom Duration (Seconds) Specifies how long it takes for the view to zoom out (using an animated transition).

This option is applicable when you use the **View in Context** functionality on the **Results** tab of the **Clash Detective** window.

View in Context Pause (Seconds) Specifies how long the view stays zoomed out.

When performing a **View in Context**, the view remains zoomed out as long as you hold the button. If you quickly click rather than press and hold the button, this value specifies how long the view stays zoomed out so that the transition is not cut in mid-flow.

Animated Transition Duration (Seconds) Specifies how long it takes to move between views.

When you click on a clash in the results grid of the **Clash Detective** window, this value is used to smooth transitions from the current view to the next.

NOTE This option is only applicable when the **Animate Transitions** check box is selected on the **Results** tab of the **Clash Detective** window.

Dimming Transparency Use the **Dimming Transparency** slider to specify the level of transparency of items not involved in clashes.

NOTE This option is only applicable when both the **Dim Other** and **Transparent Dimming** check boxes are selected on the **Results** tab of the **Clash Detective** window.

Use Wireframes for Transparent Dimming When you select this option items not involved in clashes will be displayed as wireframes.

NOTE This option is only applicable when both the **Dim Other** and **Transparent Dimming** check boxes are selected on the **Results** tab of the **Clash Detective** window.

TimeLiner Page

Use the options on this page to customize the **TimeLiner** options.

Auto Select Attached Items Indicates whether selecting tasks in the **TimeLiner** window automatically selects attached objects in the **Scene View**. This check box is selected by default.

Beginning of Working Day (24h) Sets the default working day start time.

Date Format Sets the default date format.

Enable Find Enables the **Find** command in the **Tasks** tab, which allows you to find model items related to tasks.

NOTE Enabling the **Find** command may slow the performance of Autodesk Navisworks.

End of Working Day (24h) Sets the default working day end time.

Report Data-Source Import Warnings If selecting, warning messages will be displayed if an issue is encountered when importing data in the **TimeLiner** window **Data Sources** tab.

Show Time Shows the time in the date columns in the **Tasks** tab.

CSV Page

Use the options on this page to customize CSV import/export options.

CSV File: Read Encoding Specifies the format of the text file that will be used when importing a CSV file into **TimeLiner**. Choose between **UTF-8** (for the traditional 8-bit character coding) and **Multibyte** (for larger coded character sets).

CSV File: Write Encoding Specifies the format of the text file that will be used when exporting a CSV file from **TimeLiner**. Choose between **UTF-8**, **UTF-8 with BOM** (byte-order mark), and **Multibyte**.

NOTE Data is exported in default order, without taking account of TimeLiner column order or selection.

Presenter Page

Use this page to adjust the **Presenter** options.

Profile You can adjust **Presenter** to your level of knowledge.

Select from the following options:

- **Basic** - gives you access to the basic **Presenter** features, with a limited editing functionality. This is the default option.
- **Standard** - gives you access to some of the advanced Presenter features.
- **Advanced** - gives you access to the advanced **Presenter** features, such as extra materials, lights, render styles.

Smoothed Textures Select this check box if you want textures to look smooth; this takes longer to render.

When this check box is clear, the textures render faster, but appear pixelated.

Filtered Textures Select this check box to turn on mipmapping. Using this option improves the appearance of textures in the distance.

Interactive Materials Indicates whether the applied **Presenter** materials are shown during navigation. By default, this check box is selected.

When this check box is clear, textures are not displayed during navigation. Using this option reduces the load on graphics cards, and gives less drop-out in heavily textured scenes. The materials reappear automatically when navigation stops, provided Autodesk Navisworks uses full render mode.

Interactive Lighting Indicates whether the applied **Presenter** lights are shown during navigation. By default, this check box is selected.

When this check box is clear, lights are not displayed during navigation. Using this option reduces the load on graphics cards, and gives less drop-out in heavily lit scenes. The lights reappear automatically when navigation stops, provided Autodesk Navisworks uses full lights mode.

Max Texture Size This option affects the visual details of the textures applied to geometry. Enter the desired value in pixels. For example, a value of 128 means the maximum texture size of 128 pixels x 128 pixels.

The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Max Image Texture Size Specifies the maximum size for texture images in pixels. For example, a value of 256 means the maximum texture size of 256 pixels x 256 pixels.

The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Max Background Texture Specifies the maximum size for background textures in pixels. For example, a value of 256 means the maximum texture size of 256 pixels x 256 pixels.

The higher the value, the higher the load on your graphics card, as more MB in memory is required to render textures.

Blend Transparent Textures Select this check box to use partially transparent objects. This option improves overall quality, but takes longer to render.

When this check box is clear, items with transparency of more than 50% are treated as completely transparent, and are not rendered.

Use Texture Anti-Aliasing Indicates whether anti-aliasing is used for procedural textures. Selecting this check box improves quality, but takes longer to render.

A procedural texture is a mathematically- generated image that represents natural elements such as wood, marble, granite, metal, stone, and so on. In **Presenter**, procedural textures have ball-style preview icons.

Hardware Shading Selects the hardware-accelerated shading.

NOTE This option can only be used on machines with OpenGL 1.5 compliant graphics cards.

Choose from the following options:

- **Off** - turns off hardware-accelerated shading.
- **Lighting** - only lights are shown.
- **Passive Shadows** - both lights and shadows are turned off during navigation, and automatically reappear when navigation stops.
- **Interactive Shadows** - both lights and shadows are shown during navigation.

Hardware Bump Maps Select this check box to display bump map textures during navigation. To use this option, enable **Hardware Shading** first.

NOTE This option can only be used on machines with OpenGL 1.5 compliant graphics cards.

Hardware Marble Select this check box to display hardware-rendered marble materials during navigation. To use this option, enable **Hardware Shading** first.

NOTE This option can only be used on machines with OpenGL 1.5 compliant graphics cards.

Vault Page

Use the settings on this page to display or hide the **Vault** add-in on the Autodesk Navisworks interface.

Show in User Interface Indicates whether the **Vault** add-in is shown on the Autodesk Navisworks interface. By default, this check box is not selected.

Scripter Page

Use the settings on this page to customize the **Scripter** options.

Message Level Selects the contents of the message file.

Choose from the following options:

- **User** - the message file only contains user messages (that is, messages generated by message actions in scripts).
- **Debug** - the message file contains both user messages and debug messages (that is, messages generated internally by **Scripter**). Debugging enables you to see what is going on in more complex scripts.

Path to Message File Use this box to enter the location of the message file. If a message file doesn't exist yet, Autodesk Navisworks attempts to create one for you.

NOTE You can't use variables in the file path.

Animator Page

Use the settings on this page to customize the **Animator** options.

Display Manual Entry Indicates whether the **Manual Entry** bar is shown in the **Animator** window. By default, this check box is selected.

Publish Dialog Box

Use this dialog box to adjust security settings for a published NWD file.

Title Displays the document title. The title doesn't have to be the same as the file name.

Subject Displays the document subject.

Author Displays the author of the document.

Publisher Displays the publisher of the document.

Published For Displays the party for which the document is being published.

Copyright Displays copyright information.

Keywords List of keywords that can be used to search for this document.

Comments Displays additional information about the document.

Password You can limit access to an NWD file by entering a password into this box. You are asked to confirm the password when you click **OK**.

Now anybody who wants to open this NWD file must type in the password you specified.

IMPORTANT If you forget your password, you cannot recover it from the published NWD file.

Display at Password This check box indicates whether or not the **Publish** dialog box is shown to users who are trying to open password-protected NWD files.

Selecting this check box gives users read-only access to document properties, enabling users to find out the details of the person they can request a password from.

Clear this check box if you don't want to provide the document information to users.

Expires Select this check box to set an expiration date for the published file. This activates the **Calendar** box, where you can pick the required date. After the selected date is passed, the file cannot be opened.

NOTE Evaluation copies of Autodesk Navisworks Manage 2012 can publish NWD files, however, these files have the same expiration date as the evaluation software. To remove the expiration date from an NWD file created by an evaluation copy, you must re-publish the file with a full Autodesk Navisworks Manage 2012 license.

May be Re-saved This check box controls whether users can republish this file as an NWD or not.

If this check box is clear, the review changes made to the published file can only be saved as an NWF file.

Display on Open This check box indicates whether or not the **Publish** dialog box is shown to the users as soon as they open the file.

If the file is password-protected, the dialog box is shown after the users have entered the correct password.

Embed Textures Select this check box if you want to embed all textures in the published file. This enables you to password-protect textures, including any custom or imported textures.

By default, this check box is clear. Any textures applied to the published file are saved in a folder with the same name as the published file, but with a `_Presenter_Maps` suffix. The folder is in the same location as the published file.

IMPORTANT If you do not select **Embed Textures** and intend to share the published file, you should provide the related `_Presenter_Maps` folder with textures and instruct that the recipient to place the folder in the same location as the published file to ensure viewing fidelity.

Embed Database Properties Select this check box if you want to embed all properties accessed via external databases in the published file.

By default, this check box is clear, and only the database links are stored in the published file.

Prevent Object Property Export Select this check box if you don't want to include object properties that come from native CAD packages in the published file. This enables you to protect intellectual property.

Ribbon: Output tab ➤ Publish panel ➤ NWD 

Ribbon: Application button ➤ Publish 

-  **Toolbar:** Classic user interface: **Standard** > **Publish** 
-  **Menu:** Classic user interface: **File** > **Publish**

Piranesi EPix Dialog Box

Use this dialog box to export the current view as a **Piranesi EPix** file.

NOTE This dialog box is only available for 3D models.

Browse Opens the **Save As** dialog box, and enables you to specify the location, and the name of the file you want to render to.

Size

Type Use the drop-down list to specify how to set the size of the exported image.

Choose from:

- **Explicit.** Gives you full control of the width and height (the dimensions are in pixels).
- **Use Aspect Ratio.** Enables you to specify the height. The width is automatically calculated from the aspect ratio of your current view.
- **Use View.** Uses the width and height of your current view.

Width Enables you to enter the width in pixels, when available.

Height Enables you to enter the height in pixels, when available.

Anti-Aliasing This option applies to OpenGL renderer only. Anti-aliasing is used to smooth the edges of the exported images. Select the appropriate value from the drop-down list. The higher the number, the smoother the image, but the longer they take to export. 4x is adequate for most situations.

Ribbon: **Output** tab > **Visuals** panel > **Piranesi EPix** 

 **Menu:** Application button > **Export** > **Images &**

Animations > **Piranesi EPix** 

QTVR Object Movie Settings Dialog Box

Use this dialog box to specify the number of pan frames and number of tilt frames that are used to export the QuickTime VR object movie.

NOTE This dialog box is only available for 3D models.

Pan Settings

Pan Min and Pan Max Define how far the model can be revolved (as though on a turntable).

Pan Initial Defines where you view the model from at the start (again, as though on a turntable, 0 or 360 would be the same point at the opposite side and 180 would be where the camera is now).

Pan Frame No. The number of frames to be used to revolve the model from the **Pan Min** position to the **Pan Max** position.

Tilt Settings

Tilt Min and Tilt Max Define how far the model can be tilted (backwards and forwards from its current position).

Tilt Initial Defines where you view the model from at the start (assuming you are looking at the model straight, -90 would be from the bottom and 90 from the top).

Tilt Frame No. The number of frames to be used to tilt the model from the **Tilt Min** position to **Tilt Max** position.

Pointing device: Export Rendered Image dialog box ► **Type** drop-down list ► **QTVR Object Movie** and click **OK**

Section Plane Settings Dialog Box

Use this dialog box to edit section plane settings.

NOTE This dialog box is only available for 3D models.

You can set the visibility and alignment of each section plane, select the current plane, and link visible section planes together.

Link Section Planes Enables you to link section planes together.

💡 Controls the visibility of the corresponding section plane. When the check box is selected, the section plane's effect is visible (that is, it cuts the scene). When the check box is deselected, the section plane's effect is not visible and all other controls for that plane are disabled.

Plane The number of the plane. Clicking on the plane number selects the entire row, and makes the selected plane current and visible.

Alignment Select alignment of the corresponding section plane from one of the 9 available options. See [Customize Section Plane Alignment](#) (page 459).



Click this button to re-apply a View, Line, or Surface alignment.

Ribbon: Sectioning Tools tab ► **Planes Settings** panel ► **Section Planes Settings** tool launcher

Glossary

I 8

Glossary of technical terms relating to Autodesk Navisworks Manage 2012.

Display Terminology

average frame rate This shows the current measured frame rate, averaged over the last second.

average frame time This shows the time taken to render the last frame.

average triangle rate This shows the rate at which triangles are being rendered and is a measure of how well your graphics card is working.

culling Culling is a process for determining items *not* to draw during the render of a scene. Autodesk Navisworks does a level of prioritized culling with the [drop-out](#) (page 833) method of rendering interactive scenes, but you have a certain level of control over other aspects of culling such as backface, near and far planes.

drop-out In order to maintain interactivity and guarantee a user-defined [frame rate](#) (page 833), Autodesk Navisworks only renders what it can in the fraction of a second it has. The remainder is “dropped out”, or not rendered.

Autodesk Navisworks prioritizes what is rendered and what is dropped out based on size of the item's bounding box, distance from viewer and size on screen, so only the less significant items in the scene are dropped out.

Once navigation has ceased, the scene continues rendering until all items are visible.

frame rate The frame rate is the number of frames per second (FPS) that are rendered in the main navigation window. Autodesk Navisworks guarantees a user-defined frame rate in order to maintain interactivity.

Export Terminology

These are terms specific to Autodesk Navisworks that are used in relation to exporting.

codec Codec stands for “COmpression-DECompression” and is a program that compresses and decompresses animations when creating and playing back AVI files. Codecs are installed independently of Autodesk Navisworks and are available when installed on your Windows™ system and the same codec that was used to create an AVI file is required to play it back.

File Terminology

cache files (.nwc) When any native CAD file is opened or appended, Autodesk Navisworks creates a cache file (.nwc) if the write cache option is set. When the file is next opened or appended, Autodesk Navisworks will read data from the corresponding cache file rather than re-converting the original data if the cache is newer than the original file. If the original file is altered, Autodesk Navisworks will re-create the cache file when it is next loaded. Cache files speed up access to commonly used files. They are particularly useful for models made up of many files of which only a few are changed between viewing sessions. Cache files can also be exported from some CAD applications where a native file reader is not available with Autodesk Navisworks. Cache options can be edited in the **Options Editor** (application

button  ► Options ► Model node).

external references External references (sometimes called reference files or “XRefs”) are shown in Autodesk Navisworks **Selection Tree** as an inserted group. Autodesk Navisworks looks for the externally referenced files in the same place as AutoCAD or MicroStation would.

If the **Unresolved XRef** dialog box is shown, then this link has somehow been broken and the referenced files need to be relocated to where AutoCAD or MicroStation would expect them to be.

If these XRefs are not important for the current session, then you can **Ignore** the reference and the file will load without that XRef inserted. Similarly, **Ignore All** will load the file without any unresolved XRefs.

You can also customize the options for the **DWG/DXF** and **DGN** file readers in the **Options Editor** to set whether external references are loaded or not. This gives you more control over file appending into Autodesk Navisworks.

faceting factor During an export from a CAD package to NWC format, or while Autodesk Navisworks is reading a native CAD file, decisions must be made as how a curved surface is reduced to flat facets. For most applications and file formats, you have control over the level of faceting that takes place.

All items, no matter what their size, will use the same faceting factor and so have the same number of sides to curved entities. Therefore, you need to experiment a little with different values to account for the size that these items will appear on screen.

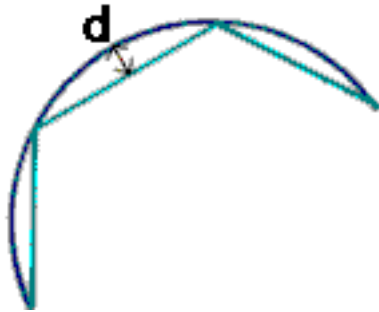
The faceting factor must be greater or equal to 0, where 0 results in the faceting factor being turned off. The default value is 1, if you double the value you get twice the number of facets, if you halve the value you get half as many facets. Larger faceting factors will result in more polygons to a model and larger Autodesk Navisworks files. There is little point having a large faceting factor if these curved entities are golf balls viewed from 200 yards!

For AutoCAD exports, the faceting factor is set from the NWCOPT command. MicroStation's faceting factor is set from **Options**, which is available from the NWCOUT export dialog.

To set the faceting factor on reading CAD files go to the **Options Editor** (click .

► **Options**, expand the **File Readers** node, and choose the relevant file reader page).

max facet deviation Maximum facet deviation is used in conjunction with faceting factor to ensure that larger objects, with too large a deviation from the original, have additional facets added. If a difference greater than the entered value is found in a model it adds more facets. The values are measured in the model units.



Where “d” is greater than the maximum faceting deviation value, more facets are added to the object.

If the max faceting deviation is set to 0, then this function is ignored and just the faceting factor is used.

published data files (NWD) Published NWD files are useful when wanting to take a snapshot of the model at a certain time. All the geometry and review information is saved into the NWD file and cannot then be changed. Published NWD files can also contain information about the file, as well as being able to be password protected and time-bombed for security. These files are also very small, compressing the CAD data by up to 80% of the original size. Published NWD files are useful when issuing models for viewing by others with the Autodesk Navisworks Freedom 2012 free viewer, as well as being appendable themselves into Autodesk Navisworks to build up a larger scene.

review files (NWF) Review files are useful when using the native CAD files appended into Autodesk Navisworks. They store the location of the appended files, along with any design reviews made in Autodesk Navisworks, such as comments, redlines, viewpoints, animations and so on.

If a group of files is appended into a Autodesk Navisworks scene, and saved as an NWF file, then on re-opening this NWF file later, once the original CAD files have been changed, the updated CAD files will be loaded into the scene for review.

shape merge threshold MicroStation shapes are polygons that can have 3 or more vertices. They’re often used to model more complex objects which can waste memory. So, Autodesk

Navisworks merges all shapes on the same level or in the same cell and with the same color into a “Shape Set” if these shapes have less than or equal to the number of vertices given by the **Shape Merge Threshold**.

Selection Terminology

These are terms specific to Autodesk Navisworks that are used in relation to selecting items.

composite objects A composite object is a group of geometry that is considered a single object in the selection tree. For example, a window object might be made up of a frame and a pane. If a composite object, the window object would be both the frame and the pane and be selected all at once.

instances An instance is a single object, which is referred to several times within a model, for example a tree. This has the advantage of cutting down on file size by not unnecessarily repeating an object.

item name The original CAD or Autodesk Navisworks assigned identifier. Any item can have a name and this name will usually come from the original CAD package that the model was created in.

item type Every item in Autodesk Navisworks has a type. Examples of types are reference files, layers, instances (sometimes called inserts), and groups. Every CAD package also has a number of geometry types, for example, polygons, 3D Solids, and so on.

selection resolution The selection resolution is the level in the selection tree you start selecting at. You can cycle through items in the tree by holding down the SHIFT key during a selection.

user name and internal name Each category and property name has two parts - a user visible string which is localized and an internal string which isn't and is mainly used by the API. By default when matching names in the **Smart Tags** and **Find Items** dialog boxes, both parts must be the same, but you can use the flags to match only on one part. You might use **Ignore User Name** if you wanted to match something irrespective of which localized version was being used.

Viewpoint Terminology

angular speed The speed that the camera moves when turning right and left in any navigation mode.

anti-aliasing Anti-aliasing improves image quality by softening the jagged edge appearance of sharp lines. 2x to 64x refers to the extra number of frames that are required for the anti-aliasing process. The greater the number of frames, the finer the effect, (with the consequent increase in rendering time).

aspect ratio Aspect ratio is the proportion of X-axis to Y-axis size. For example, in exporting a bitmap of a viewpoint, maintaining the aspect ratio would keep the proportion of the view even if the number of pixels was different.

camera-centric Navigation modes in which the camera is moved around the model.

field of view The field of view of a camera is the angle that the camera can see. A large field of view will fit more into the view, but will look distorted and a small field of view will tend to make the view more flat, tending towards an orthographic view. There are two fields of view in Autodesk Navisworks - vertical and horizontal. Editing one will change the other and the two are related by the viewpoint's **aspect ratio** (page 837).

focal point The focal point is the position in 3D space that the camera will rotate around or zoom into in examine, orbit, turntable and zoom modes.

model centric Navigation modes in which the model is moved in front of the camera.

roll The roll of the camera is its angle around the viewing axis. This cannot be edited in a navigation mode where the world up vector stays upright (walk, orbit and turntable).

saved attributes Each viewpoint can optionally save the state of its hidden and "required" items, as well as any material (color and transparency) overrides. Then, on recalling the viewpoint, those same items are re-hidden, re-made required, and the materials reinstated. This can be useful in the creation of animations when dragging on viewpoints onto an empty animation.

tilt angle This is indicated in the scene's units below (negative) or above (positive) horizontal (0) at the base of the **Tilt** window.

viewpoint up vector The direction that Autodesk Navisworks considers "up" is called the "viewpoint up vector". This is maintained in the walk, orbit and turntable modes. This may be also referred to as "world up vector".

Clash Detective Terminology

clash status Each clash has a current status associated with it and each status has a colored icon to identify them. This status is updated automatically by Clash Detective or can be manually overridden if desired. The statuses are as follows:

- **New** - a clash found for the first time in the current run of the test.
- **Active** - a clash found in a previous run of the test and not resolved.
- **Reviewed** - a clash previously found and marked by somebody as reviewed.
- **Approved** - a clash previously found and approved by someone.
- **Resolved** - a clash found in a previous run of the test and not in the current run of the test. It is therefore assumed to be resolved.

If the status is changed to **Approved**, **Clash Detective** takes the user currently logged on as the person who approved it.

Enabling hyperlinks will show clash results using the relevant status icon .

clash test status A clash test can have one of 4 statuses:

New indicates a clash test that has not yet been run with the current model.

Done indicates a clash test that has been successfully run with the latest version of the model.

Old indicates a clash test that has been altered in some way since being set up. This might include changing an option, or having loaded the latest revision of the model.

NOTE Individual clash statuses can still be edited in a clash test with a status of **Old**.

Partial indicates a clash test that has been interrupted during execution. Results are available up to the point of interruption.

clearance clash A clash in which the geometry of item 1 may or may not intersect that of item 2, but comes within a distance of less than the set [tolerance](#) (page 839).

duplicate clash A clash in which the geometry of item 1 is the same as that of item 2, located within a distance of between zero and the set [tolerance](#) (page 839). A tolerance of zero would therefore only detect duplicate geometry in exactly the same location.

hard clash A clash in which the geometry of item 1 intersects that of item 2 by a distance of more than the set [tolerance](#) (page 839).

intersection method A standard **Hard** clash test type applies a **Normal Intersection Method**, which sets the clash test to check for intersections between any of the triangles defining the two items being tested (remember all Autodesk Navisworks geometry is composed of triangles). This may miss clashes between items where none of the triangles intersect. For example, two pipes that are exactly parallel and overlap each other slightly at their ends. The pipes intersect, yet none of the triangles that define their geometry do and so this clash would be missed using the standard **Hard** clash test type. However, choosing **Hard (Conservative)** reports all pairs of items, which *might* clash. This may give false positives in the results, but it is a more thorough and safer clash detection method.

severity For hard clashes, the severity of a clash depends on the intersection of the two items intersecting. Hard clashes are recorded as a negative distance. The more negative the distance, the more severe the clash. Hard clash severity depends on whether the **Conservative** or **Normal Intersection Method** has been applied (see [intersection method](#) (page 838) for more details on this). If **Normal**, the greatest penetration between a pair of triangles is measured. If **Conservative**, the greatest penetration of space around one item into the space around another is measured.

For clearance clashes, the severity depends on how close one item invades the distance required around the second. For example, an item coming within 3mm is more severe than an item coming within 5mm of the other.

For duplicate clashes, the severity depends on how close one item is to the other. When the distance between them is zero, it is more likely that this is duplicate geometry, where as items that are further apart are more likely to be different objects and therefore have a lesser severity.

tolerance **Tolerance** controls the severity of the clashes reported and the ability to filter out negligible clashes, which can be assumed to be worked around on site. **Tolerance** is used for [hard clash](#) (page 838), [clearance clash](#) (page 838) and [duplicate clash](#) (page 838) types of clash test. Any clash found that is within this tolerance will be reported, whereas clashes outside of this tolerance will be ignored. So for **Hard** clashes, a clash with a [severity](#) (page 838) of between zero and the tolerance value will be ignored, whereas for **Clearance** clashes, a clash with a severity of more than the tolerance value will be ignored as it is further away than the distance required. Similarly, a **Duplicate** clash with a severity of more than the tolerance value will be ignored as it is likely to be a separate, yet identical piece of geometry.

Index

- 2D 213
- 2D Navigation wheel 298
- 2D/3D object associations 356
- 3D mouse devices 299
- 3D workspace 240
- 3Dconnexion
 - options 781
- 3Dconnexion 3D mouse 299
- 3DS export options 821
- 3DS options 792
- 4D 660

A

- ADN 27
- align viewpoint 240
- animation
 - action types 546
 - actions 545
 - add sets 529
 - capture 531
 - create 526
 - create scenes 527
 - cuts 472
 - delete scenes 527
 - edit 471
 - enable scripts 475, 547
 - event conditions 544
 - event types 543
 - events 543
 - export settings 737
 - object 471
 - organize scenes 527
 - play 475, 526
 - record 471
 - scope 501
 - scripts 541
 - sets 529
 - share 477
 - update sets 531
 - viewpoint 471

- Animation toolbar 123
- Animator window 502
 - options 827
 - toolbar 502
- annotations 399
- appearance definitions 632
- Appearance Profiler 437
 - options 739
- append model 222
- ARX plugin 192
- ASCII laser options 792
- Auto-Save 86
 - options 766
- Autodesk Channels 23
- Autodesk Developer Network 27
- Autodesk graphics 155
- AVI 477

B

- background effects 323
 - settings 741
- backup 86

C

- CAD applications 164
- CAD preview 196
- cameras 301
 - align 305
 - blank camera 534
 - capture viewpoints 534
 - from current viewpoint 534
 - orthographic 301
 - perspective 301
 - straighten 304
 - tilt 303
- Center tool 247
- CIP (Customer Involvement Program) 28
- CIS/2 options 793

- Clash Detective 669
 - batches 686
 - manage results 705
 - options 822
 - report results 717
 - results 703
 - rules 691
 - select items 696
- clash test types 697
- clashes 669
 - assign 741
 - reset viewpoint 708
 - run tests 703
 - soft 700
 - time-based 699
- close toolbars 115
- collaborate
 - drive 479
 - refresh 479
 - session 479
 - start 479
 - Windows NetMeeting 479
- Collaborate Bar toolbar 124
- collision 312
 - options 742
- color 388
- color-code 437
- command line 88
- comments 399
 - clash result 399
 - edit 412
 - find 414
 - IDs 420
 - review 414
 - search set 399
 - selection set 399
 - status 412
 - TimeLiner task 399
 - view 399
 - viewpoint 399
 - viewpoint animation 399
- Communication Center 15
 - settings 23
 - Subscription Center 13
- compare
 - items 365

- models 365
- compass (ViewCube) 274
- compatible CAD applications 164
- configuration files 159
- constrained orbit 271
- convert object properties 743
- create files 210
- crouching 312
- culling 327
 - area 327
 - backface 327
 - near and far clipping planes 327
 - options 744
 - require objects 330
- custom properties 369
- custom scene views 130
- Customer Involvement Program 28
- customize 745
 - commands 745
 - keyboard 126
 - navigation bar 286
 - options 746
 - toolbars 126, 745
- cuts 472

D

- database links 371
- default collision 747
- delete files 222
- developer options 776
- DGN export options 819
- DGN options 793
- display options 777
- display units 157
- Display Units options 768
- dockable windows 132
- DWF options 796
- DWG export options 813
- DWG/DXF options 797

E

- edit
 - animations 471
 - comments 412

- key frame options 748
- links 751
- redlines 413
- viewpoint options 751
- Edit menu 111
- email 227
- enable scripts 475
- environment options 766
- exit 85
- export
 - 3D DWF/DWFX 489
 - animations 494
 - clash tests 500
 - FBX 492
 - images 494
 - KLM 490
 - PDS tags 500
 - Piranesi EPix 497
 - rendered image 753
 - search criteria 498
 - search sets 498
 - simulation 664
 - TimeLiner CSV 500
 - TimeLiner image 664
 - viewpoints 498
 - viewpoints report 499

F

- face views 278
- Faro options 801
- favorites links (InfoCenter) 17
- FBX options 801
- field selector 620
- file exporters 192
 - ArchiCAD 206
 - AutoCAD 192
 - DGN 204
 - Max 205
 - MicroStation 201
 - Revit 200
 - Viz 205
- file formats
 - NWC 163
 - NWD 163
 - NWF 163

- supported CAD 168
- supported laser scan 169
- File menu 109
- file options 150, 754
 - culling 755
 - DataTools 758
 - headlight 757
 - orientation 756
 - scene lights 758
 - speed 757
- file readers 170
 - 3DS 170
 - ASCII laser scan 171
 - Bentley AutoPLANT 172
 - CIS/2 172
 - DGN 180
 - DWF/DWFX 179
 - DWG/DXF 175
 - Faro Scan 181
 - FBX 181
 - IFC 182
 - IGES 183
 - Inventor 183
 - JTOpen 184
 - Leica Scan 185
 - MAN 185
 - Parasolid 187
 - PDS 187
 - Pro/Engineer 187
 - Riegl Scan 188
 - RVM 188
 - SAT 189
 - SketchUp SKP 189
 - STEP 190
 - STL 190
 - VRML 191
 - Z+F Scan 192
- find 349
 - comments 414
 - InfoCenter information 12
 - links 428
 - objects in multi-sheet files 356
 - redline tags 419
- fly 271
- focal point 270
- focus on clash 708

- Forward tool 248
- frame rate 757
- Front view 279
- frustum culling 327
- Full Navigation wheels 296
- full screen mode 130

G

- gizmos 160
- global options 150
 - developer 158
 - display units 157
 - environment 95
 - export 151
 - import 151
 - location 154
- graduated background 323
- graphics system 155
- gravity 311

H

- hardware acceleration 333
- head-up display 306
- Help
 - access more information 25
 - concepts 18
 - displaying 18
 - print 22
 - procedures 18
 - quick reference 18
 - search 19
 - searching 12
 - training 26
- hide objects 348
- highlighting 346
- Home view 282
- horizon background 323
- HUD 306
- hyperlinks 421

I

- IFC options 802

- image export options 760
- import

- clash tests 489
 - PDS display sets 486
 - PDS tags 487
 - search criteria 485
 - search sets 485
 - viewpoints 488

- InfoCenter

- about 11
 - favorite links 17
 - searching 12
 - settings 23
 - Subscription Center 13

- InfoCenter Settings 761

- Autodesk Channels 762
 - Balloon Notifications 762
 - Communication Center 762
 - RSS Feeds 763

- installation 36

- deployment 44
 - deployment issues 77
 - general issues 71
 - languages 33
 - licensing issues 80
 - maintenance issues 83
 - multiple users 44
 - networking issues 81
 - side-by-side 43
 - stand-alone 29
 - troubleshooting 71

- interferences 669

- Inventor options 803

J

- JTOpen options 805

K

- keyboard shortcuts 142
- keyframes 538
 - capture 538
 - edit 538

L

- launch 85
- Leica options 804
- lighting 319
 - full lights 320
 - head light 321
 - no lights 322
 - scene lights 320
- links 421
 - add 427
 - attachment points 424
 - categories 421
 - control display 422
 - customize 423
 - default 430
 - delete 430
 - edit 430
 - external database 371
 - find 428
 - follow 428
 - options 771
 - reset 430
 - section plane 464
 - standard 421
 - user-defined 421
- load backup 86
- locations 154
 - option settings 766
- look around 266
- Look tool 249

M

- MAN options 805
- markups 399
- MDL plugin 201
- measuring 393
 - options 769
- Menu bar 109
- menus 109
- merge
 - files 226
 - TimeLiner data 226
- model units 223
- Model Views toolbar 122

- mouse devices 299
- move objects 383
- move toolbars 115
- multi-sheet files 213
 - add sheets/models 214
 - append 219
 - delete 215
 - find objects 356
 - merge 219
 - navigation controls 138
 - prepare 215
 - Project Browser 215
 - rename 215
- multiple-user installation 44

N

- navigation 239
 - navigation bar 284
- Navigation Bar options 782
- Navigation Mode toolbar 119
- navigation modes 264
 - Constrained Orbit 271
 - Fly 271
 - Free Orbit 270
 - Look Around 266
 - Orbit 269
 - Pan 268
 - Turntable 271
 - Walk 265
 - Zoom 267
 - Zoom Box 268
- Navigation Tools toolbar 125
- NetMeeting 479
- network deployment 44
- Network License Activation 50
- Network License Manager 50
- network share 52
- new files 210
- new link options 763
- notifications 15
- NWC 163
 - options 790
- NWD 163
 - options 790
- NWF 163

O

- object attributes 382
 - color 388
 - during animation 383
 - move 383
 - resize 383
 - rotate 383
 - snapping 389
 - transparency 388
- Object Enablers 177
- Object Manipulation toolbar 125
- open 209
- open toolbars 115
- Options Editor 764
 - File Exporters 813
 - File Readers 792
 - General 765
 - Interface 767
 - Model 787
 - Tools 822
- orbit 269
- Orbit tool 251
- orientation 240
- overlay text 630

P

- pan 268
- Pan tool 255
- Parasolid options 806
- pauses 472
- PDS options 806
- performance options 787
- pick radius 341
- Piranesi EPix export 829
- pivot points 252
- plain background 323
- Presenter 549
 - adjust materials 334
 - advanced lighting 577
 - advanced materials 565
 - archives 551
 - background effects 588
 - custom rules 602
 - edit materials 561

- effects 587
- foreground effects 593
- graphics 155
- lighting 568
- materials 556
- options 824
- position lights 569
- predefined rendering styles 596
- predefined rules 601
- rendering styles 595
- shadows 576
- texture space 598
- user archives 552
- preview 483
- primitives 325
 - lines 325
 - points 326
 - snap points 326
 - surfaces 325
 - text 327
- print 483
 - current viewpoint 484
 - preview 483
 - setup 483
- privacy 16
- product news 15
- progress bar 138
- project directory 154
- properties
 - custom 369
- publish 210
 - options 827

Q

- QTVR Object Movie settings 829
- Quick Find 355
- quick properties 432
 - options 775
- quit 85

R

- Readme 27
- receive files 227
- recently used files 95

- record animations 471
- recover 86
- redlines
 - cloud 403
 - color 403
 - edit 413
 - erase 403
 - find tags 419
 - freehand 403
 - line 403
 - line string 403
 - review 414
 - status 412
 - tag 411
 - tag IDs 420
 - tags 411
 - text 403
 - thickness 403
 - view 403
- redo 139
- reference views 307
- refresh files 225
- rename 210
- rendering 317, 553
 - accelerate 333
 - control 331
 - full 318
 - hidden line 319
 - shaded 318
 - stereo 334
 - wireframe 319
- Rendering Style toolbar 120
- reset 390
 - clash viewpoint 708
- resize objects 383
- restore 390
- review 399
- Review menu 113
- Revit export options 817
- Rewind tool 256
- ribbon 98
- Rich Photorealistic Content 583
- Riegl options 807
- rotate objects 383
- RPC (Rich Photorealistic Content) 583
- RSS feeds 23

- RVM options 808

S

- SAT options 809
- save 210
 - automatically 86
 - search sets 362
 - selection sets 362
 - viewpoints 447
- scene statistics 238
- scene view 129
- Scripter options 826
- search
 - InfoCenter 12
- search directories 159
- search sets 360
 - delete 362
 - export 362
 - import 362
 - rename 362
 - save 362
 - update 362
- section plane
 - settings 830
- section plane sets 536
- section planes
 - align 461
 - disable 456
 - enable 456
 - link 464
 - move 461
 - slider step size 461
- sectioning
 - box attributes 465
 - boxes 465
 - modes 455
 - planes 456
- Sectioning toolbar 122
- selection 337
 - commands 344
 - options 768
 - pick radius 341
 - resolution 345
 - sets 360

- selection sets
 - delete 362
 - rename 362
 - save 362
 - update 362
- Selection Tools toolbar 118
- Selection Tree 338
 - sort 338
- send 227
- share 227
 - animations 477
 - export files 489
 - import files 484
 - print 483
 - viewpoints 454
- sheet units 223
- shortcuts 142
- show internal properties 158
- side-by-side installation 43
- simulation
 - 4D 660
- simulation settings 624
- site directory 154
- SKP options 810
- snapping 389
 - options 770
- stand-alone installation 29
- Standard toolbar 117
- start 85
- status 412
- status bar 138
- SteeringWheels 287
 - options 784
- stereo 334
- STL options 811
- straighten 304
- Subscription Center 14
- SwitchBack
 - AutoCAD 434
 - MicroStation 434
 - Revit 434
- switches 88
- system requirements
 - network deployment 44
 - stand-alone installation 30

T

- tag IDs 420
- tilt 303
- TimeLiner 607
 - animation 664
 - animation to schedule 665
 - animation to tasks 667
 - appearance definitions 632
 - attach tasks 644
 - choose columns 619
 - configure 615
 - CSV options 823
 - data sources 613, 654
 - external project files 650
 - options 823
 - rules 619
 - scripts to tasks 668
 - simulate 617
 - simulation playback 617
 - synchronize tasks with schedule 659
 - task types 616
 - tasks 609, 637
 - validate schedule 649
- toolbar options 115
- toolbars 115
 - Animation 123
 - Animator 502
 - Collaborate Bar 124
 - customize 126
 - delete 126
 - Model Views 122
 - Navigation Mode 119
 - Navigation Tools 125
 - new 126
 - Object Manipulation 125
 - Rendering Style 120
 - Sectioning 122
 - Selection Tools 118
 - Standard 117
 - Workspace 120
- Tools menu 114
- Tour Building wheels 294
- transform 223
- transparency 388
- turntable 271

U

- UI 91
- undo 139
 - options 765
- unhide objects 348
- units and transform 759
 - model units 759
 - origin 759
 - rotation 759
 - scale 759
 - sheet units 759
- up direction 252
- Up/Down tool 257
- updates 15
- upgrade 43
- user interface 91
- user Interface
 - options 787

V

- vault
 - about 723
 - append files 728
 - check in 729
 - check in settings 732, 733
 - check out 727
 - create folder 734
 - default working folder 725
 - global options 826
 - launch 724
 - log in 724
 - merge files 728
 - open files 728
 - options 826
 - refresh 729
 - select file 734
 - select location 733
 - shared working folder 725
 - undo check out 730
- vault add-in 723
- view
 - fit model to scene area 272
 - fit selection to scene area 273
 - hold items 310

- set focal point 310
 - third person perspective 313
- View menu 112
- View Object wheels 292
- ViewCube 273
 - appearance 273
 - compass 273
 - lock to selection 283
 - options 783
- viewing options 754
- viewpoint defaults options 770
- Viewpoint menu 113
- viewpoints 441
 - default collision options 451
 - default view attributes 451
 - delete 449
 - edit 449
 - export 454
 - folders 448
 - organize 448
 - save 447
 - use 448
- views
 - ViewCube orientation 277
- VRML options 811

W

- walk 265
- Walk tool 258
- wheels 287
- window
 - Animator 502
 - Comments 400
 - Edit Viewpoint 449
 - Find Comments 414
 - Find Items 349
 - Find Items in Other Sheets and Models 357
 - Measure Tools 391
 - Presenter 549
 - Project Browser 215
 - Properties 367
 - Redline Tools 403
 - Saved Viewpoints 442
 - Scripter 502

- Selection Sets 360
- TimeLiner 608
- windows
 - auto-hide 132
 - dock 132
 - tile 132
 - undock 132
- Workspace toolbar 120
- workspaces 140
 - create 140
 - load 140
- save 140
- shared 140
- world orientation 240

Z

- Z + F options 812
- zoom 267
- zoom area 268
- Zoom tool 263