



TEAMCENTER

3D Model Management on Rich Client

Teamcenter 2412

Unpublished work. © 2025 Siemens

This Documentation contains trade secrets or otherwise confidential information owned by Siemens Industry Software Inc. or its affiliates (collectively, "Siemens"), or its licensors. Access to and use of this Documentation is strictly limited as set forth in Customer's applicable agreement(s) with Siemens. This Documentation may not be copied, distributed, or otherwise disclosed by Customer without the express written permission of Siemens, and may not be used in any way not expressly authorized by Siemens.

This Documentation is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this Documentation without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made.

No representation or other affirmation of fact contained in this Documentation shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

If you have a signed license agreement with Siemens for the product with which this Documentation will be used, your use of this Documentation is subject to the scope of license and the software protection and security provisions of that agreement. If you do not have such a signed license agreement, your use is subject to the Siemens Universal Customer Agreement, which may be viewed at <https://www.sw.siemens.com/en-US/sw-terms/base/uca/>, as supplemented by the product specific terms which may be viewed at <https://www.sw.siemens.com/en-US/sw-terms/supplements/>.

SIEMENS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS DOCUMENTATION INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY. SIEMENS SHALL NOT BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, CONSEQUENTIAL OR PUNITIVE DAMAGES, LOST DATA OR PROFITS, EVEN IF SUCH DAMAGES WERE FORESEEABLE, ARISING OUT OF OR RELATED TO THIS DOCUMENTATION OR THE INFORMATION CONTAINED IN IT, EVEN IF SIEMENS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

TRADEMARKS: The trademarks, logos, and service marks (collectively, "Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Marks, as applicable. The use herein of third party Marks is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A list of Siemens' Marks may be viewed at: www.plm.automation.siemens.com/global/en/legal/trademarks.html. The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

About Siemens Digital Industries Software

Siemens Digital Industries Software is a global leader in the growing field of product lifecycle management (PLM), manufacturing operations management (MOM), and electronic design automation (EDA) software, hardware, and services. Siemens works with more than 100,000 customers, leading the digitalization of their planning and manufacturing processes. At Siemens Digital Industries Software, we blur the boundaries between industry domains by integrating the virtual and physical, hardware and software, design and manufacturing worlds. With the rapid pace of innovation, digitalization is no longer tomorrow's idea. We take what the future promises tomorrow and make it real for our customers today. Where today meets tomorrow. Our culture encourages creativity, welcomes fresh thinking and focuses on growth, so our people, our business, and our customers can achieve their full potential.

Support Center: support.sw.siemens.com

Send Feedback on Documentation: support.sw.siemens.com/doc_feedback_form

Contents

Viewing 3D models

Viewing 3D models overview	1-1
Displaying 3D models	1-1
Display model parts and assemblies	1-1
Acknowledging security markings	1-3
Switch between orthographic or perspective viewing modes	1-4
Clip geometry from the 3D Viewing window	1-5
Hide parts that obscure visibility	1-6
Display information about a part or assembly	1-7
Add grid lines to the 3D Viewing window	1-8
Overview of B-Rep data	1-9
Display B-Rep vertices and edges	1-10
Save an image of a 3D model	1-11
Display and hide 3D markups and 3D GD&T markups	1-11
Pack or unpack assembly elements	1-11
Using Massive Model Visualization	1-12
Setting 3D preferences	1-17
Navigating through models	1-34
3D navigation overview	1-34
Examine your model	1-35
Return to the previous view	1-36
Move to the next view	1-36
Fit all visible parts into the Viewing window	1-37
Zoom to an area	1-37
Center a point of the model	1-37
Align the model view to the normal of a face	1-38
Rotate around the model	1-38
Rotate around a point in the model	1-38
Rotate your view to align to an axis	1-39
Rotate your view a specified number of degrees	1-41
Rotate 3D models continuously	1-42
Pan your model	1-43
Zoom around 3D models	1-43
Fly through your model	1-43
Look around your model	1-44
Return the view of your 3D model to level	1-45
Follow the terrain	1-45
Detect collisions	1-45
Explore your model	1-46
Look at an object on your model while exploring	1-47
Choose your initial position in the model	1-47
Go to a set location and position	1-48
Set a home location and position	1-48
Standard views	1-48

Use multi-touch gestures to view 3D models	1-54
Set examine navigation preferences	1-55
Set fly navigation preferences	1-58
Set explore navigation preferences	1-60
Set 3D performance preferences	1-61
Using the 3D Performance toolbar	1-67
Adjusting interactivity	1-68
Troubleshooting graphics performance	1-70
Selecting parts	1-70
Part selection overview	1-70
Select parts	1-71
Select parts in a rectangular area	1-72
Select parts in a rectangular volume	1-73
Identify selected parts	1-74
Selecting part features	1-75
Overview of selecting part features	1-75
Select part features	1-76
QuickPick overview	1-78
Previewing part feature selection	1-79
Calculating points	1-84
Managing assemblies	1-86
Overview of working with assemblies	1-86
Load and visibility statuses of assembly nodes	1-88
Specify assembly preferences	1-89
Expand and collapse the assembly	1-90
Lock or unlock an assembly	1-91
Search the assembly tree of a local .jt file	1-91
Searching assemblies saved in Teamcenter	1-93
Working with assembly columns	1-98
Working with leaf components	1-100
Working with alternate hierarchies	1-100
Viewing model layers	1-104
Working with end items	1-106
Choosing reference sets	1-108
Simplifying complex JT data	1-113
Locating assembly parts that failed to load	1-116
Creating part reports	1-118
Viewing NX advanced assembly operations	1-124
View NX part name and count information	1-126
Compare different versions of your assembly	1-126
Export node and model names	1-127
Part groups	1-128
Part groups overview	1-128
Create, sort, and reorder part groups	1-129
Create alternate hierarchies from groups	1-130
Display, hide, and select part groups	1-130
Replace parts in a group	1-131
Change the appearance of parts in a group	1-132
Return groups to their original appearance	1-133

Filtering parts	1-133
Using filters to load, display, or select parts	1-133
Create filters to load, display, or select parts	1-134
Filter types	1-135
Create compound filters	1-171
Saving filters	1-171
Save filters	1-172
Delete saved filters	1-172
Export saved filters	1-172
Import filters	1-173
Apply previously saved filters	1-173
Move filters in the Filter Sequence list	1-174
Rename filters in the Filter Sequence list	1-174
Delete filters from the Filter Sequence list	1-174
Clipping volumes	1-174
Clipping volumes overview	1-174
Clipping volume limitations	1-175
Clip areas of models	1-175
Adjust clipping volume manipulators	1-176
Hide the manipulator or the green motion jack	1-178
Prevent a part or assembly from being clipped	1-178
Remove a clipping volume	1-179
Set clipping properties	1-179
Using virtual reality devices	1-180
Interacting with your model	1-180
Using stereo viewing	1-182
Viewing JT files containing watermarks	1-187
Overview of JT files containing watermarks	1-187
Export JT watermark files	1-187
Watermarks and page extents	1-187
 Searching 3D models	
Find parts based on a property value	2-1
Search the assembly tree of a local .jt file	2-6
Search an assembly saved in Teamcenter	2-8
Create filters to load, display, or select parts	2-10
Create part reports	2-12
Display PMI types using filters	2-12
Creating visual reports	2-13
Overview of visual report expressions and rules	2-13
Create a quick visual report	2-17
Find parts based on a property value	2-18
Create a single rule visual report using local scoping	2-23
Create a visual report containing multiple rules using local scoping	2-32
Create visual reports using Teamcenter scoping	2-39
Create visual reports using Teamcenter structure lines	2-42
Create visual reports using relative dates	2-44
Comparing visual reports using snapshots	2-46

Analyzing 3D models

Comparing parts	3-1
Part comparison overview	3-1
Compare 3D parts or models	3-2
Perform side by side 3D comparisons	3-4
Specify part comparison preferences	3-4
Sectioning models	3-7
Sectioning models	3-7
Create a cross section	3-7
Create a series of cross sections	3-9
Delete cross sections	3-10
Center a point of the section	3-11
Reposition section planes	3-12
Overview of section grid options	3-23
Edit material thickness values	3-24
Overview of clipping options	3-25
Clip cross sections	3-27
Add or remove parts from a cross section	3-28
Using the Cross Section viewer	3-29
Work with cross sections in the Cross Sections view	3-31
Working with PMI cross sections	3-32
Measure cross sections	3-33
Creating point sheets	3-33
Capture cross sections as 3D documents	3-36
Capture cross sections as 2D images	3-36
Set cross section preferences	3-37
Change the appearance of a cross section	3-42
Creating cross section reports	3-44
Measuring parts	3-48
Overview of measuring parts	3-48
Measuring single part features	3-49
Measuring distances and angles between part features	3-53
Measuring between multiple features (Chain)	3-56
Measuring from one feature to multiple features (Fan)	3-58
Measuring area, volume, and mass for a set of parts	3-59
Displaying the bounding box for selected parts	3-61
Measuring between sets of parts	3-63
Measuring minimum and maximum distances	3-65
Measuring centerline curves of surfaces	3-67
Measuring relative to a coordinate system	3-69
Displaying approximate or exact measurements	3-71
Manipulating measurement display labels	3-72
Set measurement preferences	3-74
Overview of associative measurements	3-80
Displaying information about active double measurement	3-82
Logging changes to measurements	3-84
Working with measurement reports	3-85
Viewing Product and Manufacturing Information (PMI)	3-92
Introduction to viewing Product and Manufacturing Information (PMI)	3-92

Overview of the PMI view	3-93
PMI types	3-97
Display PMI in the viewer	3-101
Determining how you want PMI displayed	3-102
Working with model views	3-115
Working with PMI effectivity	3-123
Working with PMI labels	3-124
Control the appearance and behavior of PMI entities	3-126
Generate reports about measurement points and spot welds	3-134
Viewing CAE results	3-135
Viewing CAE results overview	3-135
Display CAE information	3-137
Set CAE annotation preferences and properties	3-137
Modify the model display	3-139
Probe CAE results	3-146
Annotate CAE Results	3-148
Performing surface analysis	3-151
Perform reflection analysis	3-151
Apply flow lines to surfaces	3-153
Perform curvature analysis	3-154
Apply random colors to surfaces	3-156
Manage surface analysis settings and effects	3-157
Specify surface analysis preferences	3-159
Working with visual reports	3-159
Overview of visual reports	3-159
Working with visual reports using Teamcenter embedded viewers	3-161
Creating visual reports	3-165
Load and manage existing visual reports	3-200
Examining visual report results	3-201
Save visual reports	3-205
Edit visual reports	3-206
Reset visual report results	3-207
Specify preferences for visual reports	3-207
See-Thru transparency	3-211
Working with path planning	3-212
Planning a path using Path Editor	3-212
Planning a path using Path Planning	3-222
Creating swept volumes	3-246
Overview of creating swept volumes	3-246
Swept volume performance considerations	3-246
Generate swept volumes from moving parts	3-248
Edit the input path of swept volumes	3-249
Generate swept volumes from stationary parts	3-249
Specify swept volume preferences	3-250
Save swept volumes	3-250
Inspecting JT files	3-251
What is JT Inspector?	3-251
Licensing JT Inspector Design Authority	3-253
JT Content	3-254

JT Integrity	3-254
JT Fidelity	3-254
SASIG Integrity	3-266
Inspecting JT data	3-269
Performing batch inspection of JT data	3-278
Working with JT Inspector results	3-308
Check geometry fidelity of translated JT files	3-328

Performing Clearance analysis

What is Clearance?	4-1
What is ClearanceDB?	4-6
Configuring the clearance analysis	4-8
Introduction to configuring clearance analysis	4-8
Overview of the clearance calculator settings	4-8
Set the clearance requirement	4-9
Set the clearance scope	4-10
Set the visibility action for clearance analyses in the 3D view	4-10
Clearance calculator settings	4-11
Change the precision of clearance calculations	4-26
Create images for clearance results	4-26
Performing the clearance analysis	4-29
Overview of performing the clearance analysis	4-29
Understanding penetration depth	4-30
Analysis types	4-33
Working with the analysis results	4-41
About clearance analysis results	4-41
Overview of the clearance results list	4-42
Clearance Results toolbar	4-46
Search and filter clearance results	4-47
Options for working with the results list	4-48
Using the results list Toolbox	4-53
Creating clearance filters with element attributes	4-55
Analyzing violating element pairs	4-60
Overview of analyzing violating element pairs	4-60
Display clearance violations in the Viewing window	4-61
Configure the element pair display	4-62
Adjust the visibility for a specific element pair	4-64
Display contact and penetration regions	4-65
Display penetration vectors	4-69
Display distance lines	4-71
Display cross sections	4-73
Display intersection volumes	4-75
Set clearance results preferences	4-77
Saving clearance results to a file or report	4-92
Save clearance results as a report	4-92
Using a saved list of clearance results	4-94
Managing clearance issues	4-96
Overview of managing clearance issues	4-96

Disposition clearance issues	4-96
Post clearance issues to an issues list	4-99
Communicate clearance issues by email	4-100
Create Teamcenter visual issues from clearance results	4-100
Set Clearance Manager preferences	4-102
Create a zone file	4-104
View the zones defined by a zone file	4-105
Working with variants	4-105
Understanding variant-based clearance analysis	4-105
Variant-based clearance analysis methods	4-108
Variant-based clearance analysis (client-side)	4-109
Variant-based clearance analysis (server-side)	4-111
Working with ClearanceDB	4-113
Work with ClearanceDB	4-113
Managing ClearanceDB issues in the database	4-115
Viewing ClearanceDB reports	4-116
Issue inheritance	4-118
 Transforming parts	
Overview of transforming parts	5-1
Transforming parts using the Part Transformation dialog box	5-1
Overview of transforming parts using the Part Transformation dialog box	5-1
Translating parts using the Part Transformation dialog box	5-2
Rotating parts using the Part Transformation dialog box	5-7
Scaling parts using the Part Transformation dialog box	5-10
Mirroring parts using the Part Transformation dialog box	5-12
Specify a coordinate system for transformations	5-14
Reset parts to their original position, orientation, and size	5-15
Using manipulators to transform parts	5-15
Overview of using manipulators to transform parts	5-15
Use manipulators to transform parts	5-16
Repositioning parts relative to other parts	5-22
Overview of repositioning parts relative to other parts	5-22
Use Relocate to reposition parts relative to other parts	5-23
Aligning parts	5-25
Overview of aligning parts	5-25
Align parts with other parts	5-25
Reset parts to their original positions after accepting alignment	5-28
Set alignment preferences	5-28
Transforming parts by dragging	5-29
Overview of transforming parts by dragging	5-29
Drag parts around the Viewing window	5-29
Create exploded views	5-31
 Constraining assemblies	
Overview of constraints	6-1
Constraint types	6-2

Creating constraints	6-2
Workflow for creating associative constraints	6-2
Create Touch constraints	6-3
Create Distance constraints	6-6
Create Fix constraints	6-8
Create Parallel or Perpendicular constraints	6-9
Create Angle constraints	6-12
Create Bond constraints	6-15
Creating constraints between subassemblies	6-16
Overview of assembly positioning	6-16
Simulating mechanisms using constraints	6-17
Create mechanisms to constrain components	6-18
Create a mechanism for an existing constraint	6-19
Move components by constraints	6-20
Set constraint preferences	6-21
How constraints affect motion clients	6-23
Loading constraints from a file	6-24
Load a constraint file	6-24
Merge constraint files	6-25
Saving constraints	6-26
Overview of saving constraints	6-26
Save constraints to a constraint file	6-26
Managing constraints	6-27
Change constraint distance or angle value	6-27
Overview of constraints in the Constraints view	6-27
Manage constraints in the Constraints tree	6-28
Understanding constraint status information	6-29
Display and hide columns in the Constraints view	6-32
Suppress constraints	6-32
Delete constraints	6-33
Reconcile broken constraint part references	6-33
Resolve unresolved constraint geometry references	6-34
Working with constraint labels	6-35
Overview of constraint labels	6-35
Manipulate constraint labels	6-36
Find a constraint in the Constraint tree	6-37
Viewing Degrees of Freedom	6-37
Degrees of Freedom overview	6-37
Show and hide Degrees of Freedom	6-40
Aligning parts using positional constraints	6-41
Using positional constraints to align components	6-41
Workflow for positioning components using positional constraints	6-41
 Working with coordinate systems	
Overview of coordinate systems	7-1
Create coordinate systems	7-3
Manage coordinate systems	7-5

Manipulate coordinate systems	7-7
Edit a coordinate system	7-11
Choose the Working Coordinate System	7-13
Delete a coordinate system	7-13
Align a part or assembly to a coordinate system	7-14
Align your view to a coordinate system	7-14
Set coordinate system appearance preferences	7-14
Select coordinate system features when selecting part features	7-16

Marking up 3D models

Working with 3D markups	8-1
Overview of 3D markups	8-1
Overview of anchored and floating markups	8-2
Setting 3D markup preferences	8-3
Drawing 3D markups	8-14
Manipulating 3D markups	8-47
Changing 3D markup appearance properties	8-52
Working with 3D layers	8-59
Page extents	8-65
Managing symbol libraries	8-68
Working with 3D collaborative review and markup	8-70
Working with thrustlines	8-80
Thrustlines overview	8-80
Create thrustlines	8-81
Manipulate thrustlines	8-81
Move parts along thrustlines	8-83
Adding parts to existing thrustlines	8-84
Set thrustline preferences	8-86
Working with 3D GD&T markups	8-87
Overview of 3D GD&T markups	8-87
Open the GD&T annotation editor	8-88
Create GD&T feature control frames	8-88
Create GD&T notes	8-92
Create GD&T datums	8-93
Create GD&T tolerances	8-94
Create a GD&T markup stack	8-96
Edit GD&T markups	8-97
Display and hide 3D GD&T markups	8-97
Change GD&T markup preferences and properties	8-97
Save 3D GD&T markups	8-98
Associate 3D GD&T markups to a view	8-98

Capturing outline drawings of models

Overview of capturing outline drawings of models	9-1
Create an outline	9-1
Overview of setting outline preferences	9-2
Set outline preferences	9-2

Enable outline highlighting for a single part	9-4
Override the default highlighting preferences for a single part	9-4

Working with 3D snapshots

Overview of 3D snapshots	10-1
What snapshots can capture	10-1
Capture and manipulate 3D snapshots	10-3
Apply snapshots	10-5
Merge snapshots	10-7
Scope snapshots to alternate hierarchies	10-9
Create snapshot references	10-9
Cycle through snapshots	10-10
Select visible product structure	10-10
Work with textures in snapshots	10-10
Using PLM XML Product Views to manage part visibility	10-13
Save snapshots as Product Views	10-13
Merge Product Views with a compatible JT or PLM XML assembly	10-14
Save snapshots	10-14
Specify snapshot preferences	10-15

Working with motion

Managing motion files	11-1
Overview of motion files	11-1
Creating motion files	11-1
Playing motion files	11-12
Changing part assignments	11-17
Use the Paths view	11-19
Optimize motion files	11-21
Save motion files	11-21
Converting .xmo and .res motion files	11-22
.xmo motion file specifications	11-25
Specify motion preferences	11-26
Recording movies	11-28
Overview of movie capture	11-28
Create a movie file	11-28
Create a movie file from snapshots	11-29
Set movie capture preferences	11-29

Changing part appearance

Overview of part appearance	12-1
Working with display modes	12-1
Display modes overview	12-1
Change the default display mode	12-3
Apply display modes to individual parts	12-4
Reset display modes assigned to parts	12-5
Using Quick Color Tool to change color and transparency	12-5

Overview of the Quick Color Tool	12-5
Use Quick Color to change color and transparency of parts	12-6
Use Quick Color to change color and transparency of faces or shapes	12-8
Copy color settings to other parts, faces, or shapes	12-9
Use Quick Color to restore original color and transparency	12-12
Working with True Shading	12-13
True Shading overview	12-13
Understanding True Shading	12-13
True Shading materials	12-15
True Shading global reflections	12-18
Apply True Shading global materials	12-19
Apply True Shading occurrence specific materials	12-20
Remove True Shading occurrence specific materials	12-21
Apply True Shading global reflections	12-22
Turn True Shading materials on or off	12-23
Display True Shading background images	12-23
Display True Shading floor effects	12-25
Change the True Shading floor position	12-27
Working with materials	12-28
Materials overview	12-28
Create new materials	12-29
Changing the color and shading of geometry	12-29
Using textures	12-34
Assigning and manipulating materials	12-57
Working with lights	12-65
Lights overview	12-65
Create directional lights	12-65
Create point lights	12-67
Create spot lights	12-68
Overview of light maps	12-70
Create light maps	12-71
Assign lights	12-72
Edit lights	12-73
Copy the settings of existing lights	12-74
Define default lighting	12-74
Change light color and intensity	12-75
Turn shadows on or off	12-75
Turn light sources on or off	12-76
Display light sources	12-76
Use the Direction Manipulator	12-77
Manipulate lights	12-77
Working with the appearance palette	12-78
Overview of the Appearance Palette	12-78
Assign appearance attributes from the Appearance Palette	12-79
Load images for use with textures	12-83
Filter Appearance Palette entries	12-83
Reuse appearance palettes with other models	12-84
Specify Appearance Palette preferences	12-86
Specify appearance reference files	12-88

Use the assembly to control appearance attributes	12-89
Overview of the assembly appearance icons	12-90
Insert 2D images into 3D environments	12-91
Apply a color range to a model	12-92
Specify appearance preferences	12-94

Generating reports

Generating reports overview	13-1
Specify content that appears in the report	13-1
Define the report header	13-2
Define report columns	13-2
Define report filters	13-3
Define a report footer	13-4
Preview reports	13-5
Specifying report output and presentation	13-5

Editing parts

Part simplification overview	14-1
Overview of the Part Working Set	14-1
Define the Part Working Set	14-4
Manage part names and LODs	14-5
Managing LOD labels	14-7
Overview of B-Rep face orientation	14-8
Reverse B-Rep face orientation	14-8
Overview of part tessellation	14-9
Tessellate parts	14-10
Overview of how to simplify parts based upon visibility	14-11
Simplify parts based upon visibility	14-12
Overview of how to reduce the part triangle count	14-15
Remove triangles	14-16
Use the Visibility and B-Rep filters	14-17
Specify Part Edit Preferences	14-19

Undoing actions

Overview of undoing actions	15-1
Undo the previous action	15-2

1. Viewing 3D models

Viewing 3D models overview

When you open 3D model files, two types of information are available: a textual representation of the model assembly and a visual representation of the geometry. The assembly appears in the **Assembly** view. The geometry appears in the Viewing window.

When you open a DirectModel (JT) assembly file that consists of more than one part, the model assembly is loaded, but the geometry is not automatically displayed in the Viewing window. This is because assemblies often consist of hundreds or even thousands of parts, and it often takes a long time to load the geometry of such complex models. Because only the assembly is initially displayed, you can turn on only the parts you want to view, and preserve system resources. When you open a model that contains a single part, the part is loaded and displayed immediately.

Note:

To enable .jt file viewing using Base Lifecycle Visualization, you must select **3D File Support** from the list of Base features during installation.

Displaying 3D models

Display model parts and assemblies

Display or hide entire models, subassemblies, or individual parts.

Note:

When you open a model that consists of a single part, the part geometry loads and the part appears in the Viewing window.

When you open a model with more than one part, the assembly information appears in the assembly tree, but part geometry does not initially load and the model does not appear in the Viewing window. You must choose which parts to view.

Tip:

When working with very large assemblies, do not load the entire model if you only need to view a few parts. After you open the model, you can expand the assembly tree and load just the parts you need to view.

Do any of the following:

To	Do this
Display or hide the entire model	- Do any of the following: - In the Assembly view, select or clear the assembly check box. - Choose View→Visibility→All On or All Off. - Right-click an empty area of the viewing window and choose All On or All Off. Note: If model geometry displays in the Viewing window after selecting All Off, turn off the active model view in the PMI tree, as this will cause the PMI part geometry to display in the Viewing window. Note: Loading large models can take a long time to complete. You can press either Ctrl-Break or Ctrl-Fn-B to stop loading the files. Note that server or FMS/FCC transactions are not interruptible.
Display or hide subassemblies	In the Assembly view, expand the assembly list until you see the name of the subassembly and then select or clear the subassembly check box.
Display or hide individual parts	- Do any of the following: - In the Assembly view, expand the assembly list until you see the name of the part and then select or clear the part check box. - In the Viewing window, right-click a part and choose View Only or Turn Off. - In the Viewing window, right-click one or more selected parts and choose Selected Only.
Use the assembly to display selected parts only	- Choose View→Visibility→Control by Selected. - In the Assembly view, select the name of a part or assembly that you want to display. All parts that are not selected are hidden. When you select parts and assemblies in the assembly, the selected parts are displayed and the deselected parts are hidden. Tip: Press the Ctrl or Shift keys to select multiple parts or assemblies.
Display all parts that you have made visible during the current work session	Choose View→Visibility→Visited On. -or-

To	Do this
	Right-click an empty area of the Viewing window and choose Visited On. Note: <i>Visited</i> parts are parts that you have explicitly made visible during the current work session. Choosing Visited On turns on all of the parts that you have displayed at any point during the session.
Swap the visibility state of parts that you have made visible during the current work session	Choose Menu → View → Visibility → Invert Visited. -or- Right-click an empty area of the Viewing window and choose Invert Visited. Note: <i>Visited</i> parts are parts that you have explicitly made visible during the current work session. Choosing Invert Visited simultaneously hides all visited parts that are currently displayed and shows all visited parts that are currently turned off.
Undo a part visibility action	Press Ctrl+Z. Tip: To reverse an undo, press Ctrl+Y.
Zoom in the camera onto selected parts	(Optional) For smooth camera transitions in the Viewing window, use the 3D Viewing preferences to turn on View interpolation. - In the Viewing window, right-click a part and choose Fly to Part. Tip: To return to the previous position, without making any changes to model position, click Fly to Part again.
Center the camera on select parts	In the Viewing window, right-click a part and choose Seek .

Acknowledging security markings

When a part or assembly component has an NX-authored security marking and you display the part in the viewer, an acknowledgment dialog box appears. Once you select the option to acknowledge the marking, the dialog box does not appear again for the part within the current session or lifetime of the part's document.

If multiple marked parts are displayed simultaneously, a separate acknowledgment is required for each unique part.

If an assembly component has a security marking, its acknowledgment supplants any markings on any part or assembly components below the marked assembly, so repetitive acknowledgments are not required.

Note:

Since the security markings are authored into the JT file, you cannot bypass the markings.

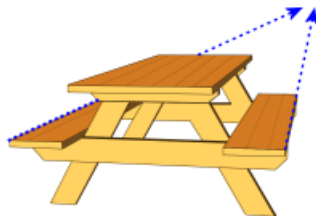
Three types of security markings can be stored in a part file in NX:

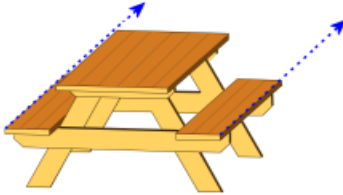
- Government security information
- Company proprietary information
- Export control

Switch between orthographic or perspective viewing modes

Display 3D models in the *Perspective* or *Orthographic* viewing modes.

- Choose **View**, and then select one of the following options:

Use this option	To do this
Perspective	Display models realistically, with depth and proportion. In Perspective viewing mode, parallel lines converge in the distance. Two parts of the same size appear to be of different sizes if one is much closer to the view point.  Perspective Note: You can adjust the field of view for the perspective viewing mode.
Orthographic	Display the exact spatial relationships among parts, regardless of their relative distances from the view point. In Orthographic viewing mode,

Use this option	To do this
	parallel lines remain parallel, even in the distance. Two parts of the same size appear the same even if one is much closer to the view point.  Orthographic Caution: Orthographic viewing is not recommended when navigating in Fly or Explore modes. It is important to see a view with depth in these modes.

Clip geometry from the 3D Viewing window

You can add a near and a far clipping plane to the 3D Viewing window.

Example:

The position of the clipping planes in front of the camera remains constant as you navigate the scene. When visualizing a large 3D model such as a factory or a ship, you can automatically clip geometry from your path as you explore the model. As the camera moves forward, geometry is clipped from the view to expose areas of the model that are otherwise obscured.

1. On the **3D Standard Views** toolbar, click **View Clipping** .
2. In the **Clipping Planes** dialog box, do the following:

To	Do this
Add a clipping plane to the near side of the viewing window	Click the Near check box, and then adjust the slider bar to position the clipping plane.
Add a clipping plane to the far side of the viewing window	Click the Far check box, and then adjust the slider bar to position the clipping plane.

Note:

The slider bar range for the **Near** and **Far** clipping planes is calculated each time the camera is moved. When the camera position changes, the positions of the slider bars in the **Clipping Planes** dialog box also change, but the positions of the clipping planes in the viewing window do not move.

3. If you are in the **perspective** viewing mode and want to expand or contract the field of view, adjust the **Field of View** slider.
4. To save the position of the clipping planes and the field of view as the default view clipping settings, click **Set as default**.

Note:

- Only the clipping plane positions and the field of view are saved as defaults, not the clipping plane state (on or off). When you turn on view clipping in a subsequent work session, the default clipping plane positions and field of view are automatically applied to the viewing window.
- If a default clipping plane position is out of range for the current view, when the clipping plane is turned on the application disregards the default and positions the clipping plane according to the size of the current model.

5. (Optional) Capture the view clipping state in a **snapshot**, session file, or PLM XML.

Note:

- To include the view clipping state in your snapshots, you must turn on the **Camera Orientation snapshot capture preference**.
- To include snapshots in PLM XML, you must turn on the **Current State of Document** option in the PLM XML export options.

Hide parts that obscure visibility

Hide parts that obscure visibility in your 3D model by making the obscuring parts transparent.

1. Select the obscured part.
2. On the **3D Visibility** toolbar, click **Hide Obscuring** .
3. In the **Hide Obscuring Parts** dialog box, move the slider to the left.

-or-

In the box, type a value between **0** and **1**.

Note:

A transparency value of 1 is invisible; a transparency value of 0 is opaque.

4. Apply the changes in one of these ways:
 - Click **OK** to apply the changes and close the dialog box. The parts that are currently hidden or transparent remain hidden or transparent as you rotate the model.
 - Click **Apply** to apply the changes and leave the dialog box open. As you rotate the model in the Viewing window, part visibility changes so that as parts move to obscure the selected part, the obscuring parts are hidden.

Display information about a part or assembly

Display the properties of a part or assembly to view information that is included in the file about the geometry, density, extent, and attributes of the part.

1. Right-click the part or assembly and choose **Properties**.
2. In the **Properties** dialog box, view any of the following information:

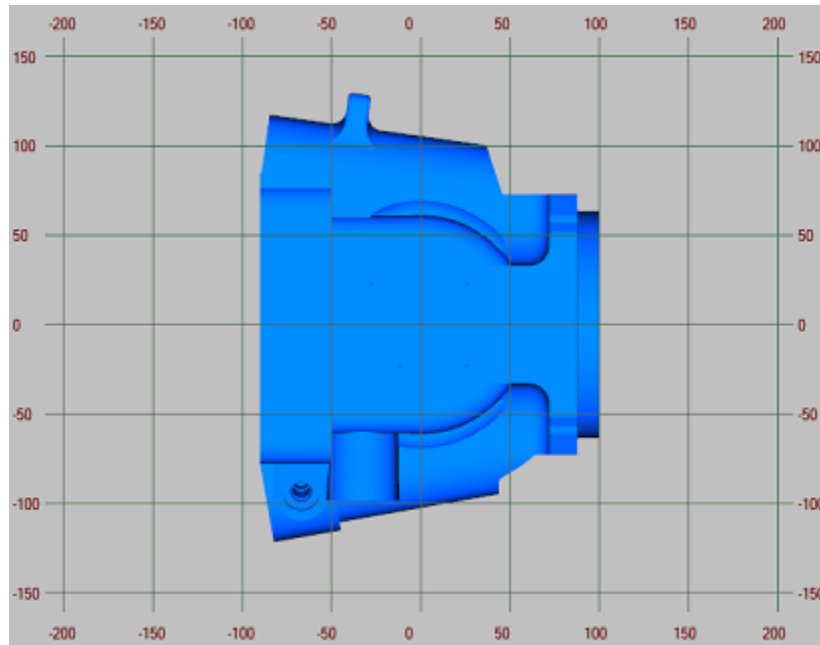
This tab	Displays information
Attributes	Metadata associated with the part or assembly. Note: If you opened the part or assembly from Teamcenter, attributes from both Teamcenter and NX appear, if present in the dataset. You can set the attributes that appear. Tip: - To view the entire attribute value, do one of the following: - Double-click the attribute or its value. - Right-click the attribute and choose View Attribute Value. - To add an attribute as a sortable column to the Assembly view, select an attribute, and click Add as Column.

This tab	Displays information
	- To print attribute and value information for the part or assembly you selected, click Print. - To save the information to a file, click Export to file.
Geometry	The values for the following Geometry Properties (Area, Volume, Density, Mass, Center of Mass) are precise measurement values if the CAD properties exist. Otherwise, the values are calculated. Tip: - Refer to the legend at the bottom of the Geometry page to determine if a value is calculated or based on metadata. - The measurement and mass units used are based on Display Units that you set from the Measurement Preferences dialog box, Calculations page. Note: For an accurate volume on a part, the following conditions must be met: - The seams on the part must be closed. If the part is not completely closed, the volume does not exist and cannot be displayed. - The triangles in the part must be consistently oriented. If you have triangles that face into a part, then the volume displayed is not accurate.
Extents	Minimum point, maximum point, center point, and length of the part along each axis.

3. Click **OK**.

Add grid lines to the 3D Viewing window

You can add grid lines to the 3D Viewing window to help you visualize the relative size of 3D data parts and assemblies. To help you visualize smaller differences, use emphasis lines. You can control emphasis line thickness, color, and spacing to improve viewing size differences.



1. Choose **View**→**Orthographic**.
2. To set axis-aligned views, choose **View**→**Standard Views** and then select **-X**, **-Y**, or **-Z**.
3. In the Viewing window, right-click and choose **Preferences**.
4. In the **Preferences** dialog box, click the **Grid** tab and select **Show WCS reference grid in Orthographic, axis-aligned views** to activate grid lines.

Note:

- You can capture and view snapshots containing grid lines as long as the grid is enabled. If you make the grid unavailable, the grid lines are not displayed.
- Grid line features are modified by navigating to the **Preferences** dialog box and then selecting the **Grid** tab.

Overview of B-Rep data

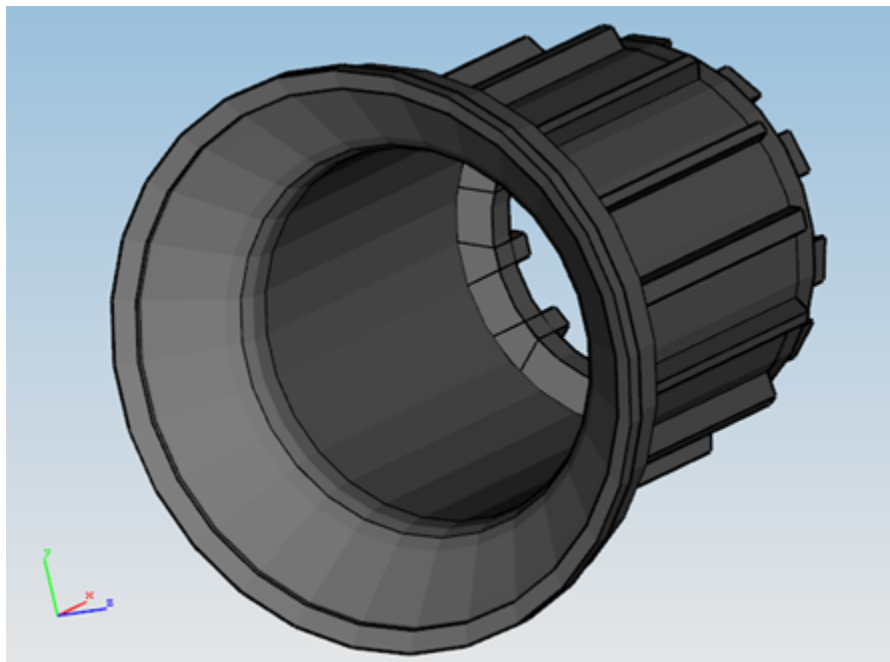
Boundary Representation (B-Rep) data gathers geometric shapes into logical features by collecting connected surface faces, edges, and vertices that emphasize the boundaries between solid and non-solid areas of a 3D model. When you open a 3D model file that includes B-Rep data, the model geometry information is converted to tessellated data, which uses primitives. However, the original B-Rep data is preserved in your model file. Compared to tessellated approximations, B-Rep data more precisely reflects the original CAD program measurements.

Note:

- Not all 3D model files include B-Rep data. For information on exporting models using B-Rep data, consult the documentation for your CAD application.
- Your geometry must include B-Rep data if you want to display vertices on your model.
- To show only B-Rep edges when using the **Draw feature lines** or **Show Edges** options, on the Display page of the 3D Viewing preferences dialog box, ensure that **With silhouette lines** is not selected, select **Use B-Rep if Available**, and for **Feature Angle** enter a value of 180.
- Standard .jt files may use either of the following B-Rep schemes:
 - JT B-Rep, a visualization format exclusive to .jt files.
 - XT B-Rep, the Parasolid format used by Solid Edge and other CAD applications.
- Parts must have B-Rep data for you to use part feature selection preview.

Display B-Rep vertices and edges

If your 3D model file includes B-Rep data, you can display the original B-Rep vertices and edges of each part.



On the **3D Display Modes** toolbar, click **Draw feature lines** .

-or-

Right-click the parts and choose **Show Edges** or **Show Vertices**.

Save an image of a 3D model

Capture and save an image of a 3D model in a 2D raster format.

- 1. Display the model parts to include in the view.
- 2. Navigate the model until the Viewing window displays the view you want to save.
- 3. Choose **Actions→Image Capture**.

A 2D image of your model appears in your Viewing window.

- 4. Save the image as a .bmp, .tiff, .png, or .jpg file.

Display and hide 3D markups and 3D GD&T markups

You can display, hide, and rearrange 3D markups and 3D GD&T markups.

To	Do this
Display or hide individual 3D layers, 3D markups, or 3D GD&T markups.	In the Assembly view, in the Markup Layers list, select or clear the check box to the left of the markup layer or markup name  .

Pack or unpack assembly elements

You can use the *packing* action to group similar elements in an assembly that was opened in Lifecycle Visualization from either Active Workspace or rich client. Packing results in a shorter and simpler structure display. This action groups identical elements in a single level of an assembly, all of which satisfy the packing criteria configured using the **BOMExcludeFromPackCheck** and **BOM_Additional_Packing_Criteria** preferences. The packed elements are represented using a single row in the structure. A numeric value is appended to this row to indicate the number of packed items.

The following apply:

- If the **Show structure packed** preference on the **3D Loader Preferences** dialog box is selected, when you open a structure for the first time, Lifecycle Visualization will attempt to automatically pack any items that can be packed based on the packing criteria settings.

- If packed elements have unmatched associated attributes, **Packed Notes** displays in the associated attribute column in the assembly tree and in the **Attribute Properties** dialog box, instead of the actual attribute value.
- When a sibling belonging to a packed node is selected in the Viewing window, the associated line item in the assembly tree reflects the highlight color (light blue) instead of the item select color (dark blue).
- An assembly, or assembly node, must have been expanded in the assembly tree before it can be packed.
- Structure lines are considered siblings and can be packed if they match the defined packing criteria *and* have the same parent within the structure.
- Available with Teamcenter Lifecycle Visualization 14.3 or higher.

To	Do this
Pack or unpack all applicable assembly nodes together with their siblings.	Choose Assembly tab→Assemblies group→Pack All  or Unpack All . This method is unavailable in Base. -or- Right-click in the assembly tree in the Project Workspace window and choose Pack All  or Unpack All  from the drop down list. -or- Choose Menu→Assembly→Pack All  or Unpack All .
Pack or unpack a specific node.	Right-click the node in the Project Workspace window and choose Pack or Unpack from the drop down list.

Using Massive Model Visualization

Overview of Massive Model Visualization

Massive Model Visualization (MMV) is a visualization technology that uses Visibility Guided Rendering (VGR) to increase performance and scalability for the viewing of extremely large 3D models, such as cars, airplanes, and ships. Models of this size typically consist of a massive amount of geometry arranged in a relatively compact space, with a huge amount of internal geometry hidden behind the “outer shell” of the product. It can take hours to display such models in their entirety, because every piece of geometry in the model needs to be retrieved and processed, which far exceeds the capability of today’s commonly available hardware. MMV technology resolves this problem by leveraging VGR techniques to load only those parts required to render a given scene; parts that are not visible because they are occluded by other parts in the foreground are not loaded. As a result, large 3D models become visible in a fraction of the time previously required.

Note:

- Visualization of MMV data requires an additional license and is supported only in the standalone viewers and the Lifecycle Viewer within the Teamcenter rich client. If you do not have the necessary license or if you load an MMV dataset in an unsupported viewer, the full model loads as standard JT data.
- MMV requires a graphics card that fully supports OpenGL and cannot be used with a remote desktop or virtual machine.

Features supported with MMV data

Features supported with MMV data include the following:

- 3D navigation tools
- Part selection options
- 3D comparisons
- Visual reports
- Filters
- Snapshots
- Image capture
- Session save

Note:

Some of the viewer features available for the visualization and analysis of JT models are not available for use with MMV data. When you open an MMV model in the standalone viewer or the Lifecycle Viewer, features that are not supported with MMV are disabled. If you want to work with an unsupported feature, you must **disable the MMV data** and load the JT files for the entire model.

Harvest MMV data from JT models

An MMV dataset consists of a spatial hierarchy of a model, which is harvested from JT data using the Teamcenter Dispatcher **mmvindexgen** translator. You can use the My Teamcenter **Translation** menu to automate the generation of the spatial index on a recurring basis, in order to capture design changes over the course of the product lifecycle.

Siemens Digital Industries Software recommends that you set the translator to run daily, during a time of non-peak Teamcenter user activity. The performance of the harvester depends on many factors, such as model size and database speed. In determining the frequency in which to run the harvester, you should determine the amount of time that it takes to run. For example, if it takes more than one hour to run, you should not set it to run every hour.

Note:

If you want to verify the translator is working properly, you do not need to run Dispatcher to see if MMV is running correctly. Just open the *Module\Translators\mmvindexgen* directory in a command window and run the *mmvindexgen.bat* file from the command prompt. For more information about running the translator, run *mmvindexgen.bat -help* instead.

1. Ensure that the Teamcenter **mmvindexgen** translator is installed and properly configured.
2. In My Teamcenter, select the item revision containing the JT model from which you want to generate MMV data.
3. Choose **Translation→Translate**.
4. In the **Translation Selection** dialog box, select the **mmvindexgen** service, and then click **Next**.
5. In the **Translation Arguments** section, configure the revision rule.
6. In the **Priority and Time Properties** section, specify the appropriate values for translation time, priority, and translation repeating schedule options.
7. Click **OK** to start the translation.
8. (Optional) Choose **Translation→Administrator Console - All** to see the progress of the translation.

After the translation is complete, the spatial index file appears in the item revision.

If the **mmvindexgen** translator fails when running under Dispatcher, consult the log file that is produced from *harvest_mmv_index*. Once you have corrected the error, run the translator on the command line to ensure it runs properly.

Manually generating MMV data

To use MMV without the Dispatcher, the MMV data must be manually generated prior to viewing new or updated assemblies in Teamcenter.

1. Load the assembly into Teamcenter.
2. Add the **Spatial Rendering** relation by doing the following.

- a. Choose **Edit→Options**.
 - b. Expand the **Options** tree and expand the **General** folder.
 - c. Select the **Item Revision** node.
 - d. On the **General** options tab, choose **Spatial Rendering** from the **Available Relations** list.
 - e. Click **+** to move it to the **Shown Relations** list.
 - f. Click **Apply**.
 - g. Click **OK**.
3. Open a Teamcenter command prompt:

From the **Start** menu, choose **Programs→Teamcenter**, and open the command prompt.

4. In the command prompt, run the **harvest_mmv_index.exe** to create the MMV data without the use of the Dispatcher.

Example:

```
C:\apps\PLM\Tc2412\tc_root\bin>harvest_mmv_index.exe -u=Tc-admin-user
-p=password -g=group -item=assy -rev=engine -rev_rules="Latest
Working"
```

This creates the SpatialTree object for the assembly.

5. **Specify to use MMV data in the viewer.**
6. **View your MMV data.**

Specify to use MMV data in the viewer

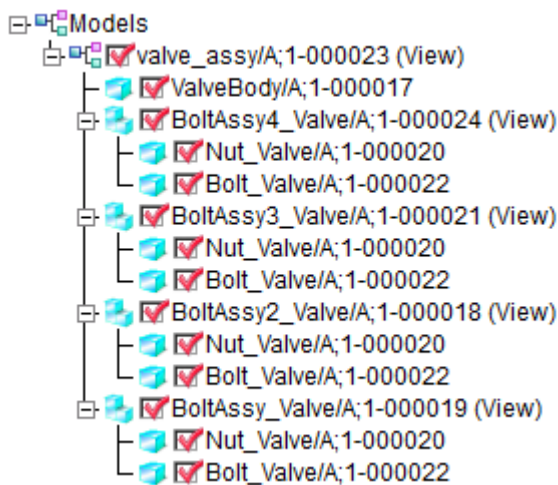
Before loading an MMV model in the standalone viewer or the Lifecycle Viewer, you must specify to use MMV data in the 3D Loader preferences.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, click the **Other** tab.
3. In the **General** section, select the **Use MMV if available** check box.

Viewing MMV data

Complete the following steps to load your MMV data once the data has been **generated**. Data does not need to be generated each time you view the MMV data. You only generate data for new MMV assemblies or when the data set has changed.

1. In the standalone viewer or the Lifecycle Viewer, ensure the **3D Loader preferences** are set to load MMV data.
2. In My Teamcenter, identify the item revision containing the MMV model you want to load, and then do one of the following:
 - To open the model in the standalone viewer, select the item revision, and then, on the **My Teamcenter** toolbar, click **Start/Open in Lifecycle Visualization** .
 - To open the model in the Lifecycle Viewer, right-click the item revision and choose **Send To→Lifecycle Viewer**.
3. In the assembly tree, select the check box for the top-level node to display the model.



4. If you want to ensure that you are viewing the most recent model geometry, **disable the MMV data**.

Disable MMV data

The product structure and geometry of an MMV model are not always in sync. The product structure for the model reflects the most current design changes, but the MMV spatial index may be out of date if the product is updated after the last translation. To obtain the latest version of the model, disable the MMV data.

To disable MMV data and load JT files for the model, do any of the following:

To disable MMV data	Do this
For the entire model	On the 3D MMV toolbar, click Exit MMV  .
	Note: Not all viewer features support MMV data. If you want to work with an unsupported feature, you must disable the MMV data and load the JT files for the entire model.
For all visible parts	On the 3D MMV toolbar, click Disable MMV visible  .
For selected parts or subassemblies	- In the assembly, select one or more parts or subassembly nodes. - On the 3D MMV toolbar, click Disable MMV selected .

Setting 3D preferences

Set 3D viewing preferences

Change the 3D viewing preferences to customize the scene in which your 3D model appears and to adjust interactively.

- Open a 3D model.
- In the Viewing window, right-click and choose **Preferences**.

The **Preferences** dialog box has four tabs, **General**, **Display**, **Grid**, and **Selection**.

- On the **General** tab, change any of the following general 3D viewing preferences:

In this section	Use this option	To
Model	Two-sided lighting	Light both faces (or sides) of polygons in your 3D model view. Tip: Turn on this option if you have parts or polygons that unexpectedly appear black.
	Draw trihedron	Show the orientation of the Global coordinate system X, Y, and Z axes. 

In this section	Use this option	To						
	View interpolation	Transition smoothly as the view changes, as if slowly moving to another position, when you do one of the following: • Apply snapshots that contain different camera orientations. • Right-click a part and choose Fly to Part. Tip: If your model contains many parts, view interpolation may not appear to be smooth. You can make the interpolation process seem smoother by setting a fixed frame rate, on the Rendering tab of the Performance dialog box.						
	Ambient light factor	Type a value between 0 and 1 to adjust the ambient light that illuminates your model. The change may be subtle. Note: When in True Shading, any changes to the ambient light value will revert back to the previous value when True Shading is turned off.						
	Anti-aliasing	Apply anti-aliasing to geometry in the 3D Viewing window. Anti-aliasing allows more than one sample point per pixel and can help edges appear less jagged. Tip: Anti-aliasing changes only apply for new windows. To create a new window, choose Window→New Window.						
	No. of samples	Choose the number of samples for anti-aliasing. A higher number of samples results in smoother edges. Anti-aliasing does not always use the specified number of samples. The graphics driver may decrease the sample number if there is insufficient video RAM available. The current number of samples being used in the Viewing window is displayed on the right side of the status bar. Mem: 191.8M/1700.0M AA:16 (H/W)						
	Set as default	Save the current Model preferences in this section as default values.						
Window	Background	To choose the display scheme for color, select one of the options below. You can choose colors for each of the four corners of a 3D window. The background color fades between the corner colors. <table> <tr> <th>Choose</th><th>Then select</th><th>To</th></tr> <tr> <td>Solid</td><td>The color from the single color list</td><td>Set the single background color that is displayed behind your 3D model in the Viewing window. (This gives the same result as</td></tr> </table>	Choose	Then select	To	Solid	The color from the single color list	Set the single background color that is displayed behind your 3D model in the Viewing window. (This gives the same result as
Choose	Then select	To						
Solid	The color from the single color list	Set the single background color that is displayed behind your 3D model in the Viewing window. (This gives the same result as						

In this section	Use this option	To
		choosing the same color for all four corners.) Vertical The fade colors from the top and bottom color list. Fade the top color to the bottom color. Horizontal The fade colors from the left and right color lists. Fade the left color to the right color. Corners The fade colors from the four color lists. Fade color between the four corner colors.
	Set as default	Set the window background options as default. Note: If you choose this option, the display preferences affect the current Viewing window as well as any new Viewing windows in the open session or future sessions. If you do not choose this option, the display preferences only affect the current Viewing window.

4. On the **Display** tab, change any of the following 3D display preferences:

In this section	Use this option	To
Model	Shaded	Show geometry in the shaded display mode.
	Tessellation Lines	Show geometry in the tessellation lines display mode, making the model appear in wireframe. You can choose the color for tessellation lines.
	Feature Lines	Show geometry in the feature line display mode. You can choose the color for feature lines.
	Invisible	Make hidden lines invisible within the model.
	Ghosted	Make hidden lines appear faint within the model Note: Shaded display mode must be off to see hidden lines.
	Visible	Make hidden lines visible within the model. Note: Shaded display mode must be off to see hidden lines.
	Construction Geometry	Show wireframe construction geometry. Always view the wireframe construction geometry On Top of the shaded geometry.

In this section	Use this option	To
	See-Thru Solids	Override other display modes and display all solid geometry as semi-transparent. This display mode facilitates viewing wireframe and point geometry.
	Transparency	Specify the transparency level when using See-Thru Solids display mode. This setting also affects parts rendered transparent with: • Section clipping. • Clipping volumes. • Visual reports. Note: Transparent parts are considered <i>unshaded</i> when picking through unshaded parts.
	Global Color	Specify the global transparency color when using See-Thru Solids display mode. This setting also affects parts rendered transparent with: • Section clipping. • Clipping volumes. • Visual reports. Note: Transparent parts are considered <i>unshaded</i> when picking through unshaded parts.
	Per Part Color	Use the part's original color as the transparency color when using See-Thru Solids display mode. This setting also affects parts rendered transparent with: • Section clipping. • Clipping volumes. • Visual reports. Note: Transparent parts are considered <i>unshaded</i> when picking through unshaded parts.

In this section	Use this option	To
Feature Lines Options	Use B-Rep if available	When generating feature lines use B-Rep data if available.
	With Silhouette Lines	Display edges that are formed by two faces with normals pointing in opposite directions relative to the view normal. In effect, this option creates a rough outline that makes the feature line display more complete. Note: Rendering with silhouette lines may cause artifacts resembling tessellation lines on thin parts or parts containing back facing triangles.
	Line Width	Specify the width of silhouette and feature lines. Enter a value between 0.1 and 4.
	Feature Angle	Specify the maximum angle between two triangles for inclusion as a feature line. Enter a value between 0 and 180. The default value is 18.
	Set as default	Save the current Display tab preferences as default values.

5. On the **Grid** tab, change any of the following 3D Viewing window grid preferences:

In this section	Use this option	To
	Show WCS reference grid in Orthographic, axis-aligned views	Display or hide the reference grid. When this option is cleared, all other options are unavailable. Note: This program uses the World Coordinate System (WCS) to plot the grid lines.
Grid Lines	Spacing	Specify the spacing between grid lines and the unit of measure. For example, entering 2 and meters creates grid lines that are 2 meters apart.
	Color	Choose a grid line color from the list.
	Pattern	Specify the grid line style.
Emphasis Lines	Show emphasis lines	Display or hide emphasis lines and select the frequency of the grid lines that you want emphasized. For Emphasis every , enter a positive number. Example: If you set Spacing at 1 meter, entering 5 creates emphasis lines every five meters.
	Color	Specify the color of emphasized lines.
Labels	Show labels	Display or hide grid line labels.

In this section	Use this option	To
	Size	Specify the label size.
	Style	Specify a type of grid line label. • Normal – grid lines are labeled with the precise unit of measure. For example, if you set spacing at 100mm on the X-axis, grid lines are plotted and labeled 100mm, 200mm, etc. • Concise – grid line labels are shortened to 1X, 2X, and so on along the X-axis and 1Y, 2Y, and so on along the Y-axis, and so forth.
	Text color	Select a font color from the list.
	Fill color	Select the background color for the label.
	Set as default	Save the current Grid preferences on this tab as default values for new Viewing windows.

6. On the **Selection** tab, change any of the following selection preferences:

In this section	Use this option	To
Selection	Parts	Indicate parts that are selected in the Viewing window by changing the Material color or by enclosing the parts in a Bounding box .
	Color	Choose the color of the material or bounding box that indicates part selection.
Selection Preview	Display Highlights	Highlight part features as you move the cursor over parts
	Display Geometric Types	Display icons that indicate the types of part features as you move the cursor over parts.
	Display Construction Lines	Display part feature construction lines as you move your cursor over parts.
	Points	Specify how to display points when highlighting part features. Choose Dot , Square , Filled Square , Diamond , Filled Diamond , Circle , Filled Circle , Plus , Cross , or Asterisk .
	Color	Specify the color used to highlight part features.
QuickPick	Show QuickPick dialog after	Specify the time delay before displaying the QuickPick prompt.
	Number of Features per type	Specify the maximum number of each type of feature to display in the QuickPick dialog box.
	Set as default	Save the current Selection preferences on this tab as default values for new Viewing windows.

Managing the JT Level of Detail (LOD)

Introduction to managing LOD

You can control access to the Level of Detail (LOD) present within .jt files. An LOD Access Profile is a filter that lets you access LODs that match the criteria of the profile you select. This enables different groups in an organization to work with LODs specific to their purposes while sharing the same .jt files.

Create an LOD Access Profile

Create an LOD Access Profile by specifying access strings that are compared to **LOD labels** within the file. For older .jt files that do not contain LOD labels, you can specify default labels for equivalent LODs in the file.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **Use LOD Access Profiles when opening documents**.
3. In the **Profile** section, click **Create**.
4. In the **Create New Profile** dialog box, in the **Profile Name** box, type the text that you want to associate with the profile.

Note:

- The default text in the input sections of this dialog box is intended to guide you through the process of completing your LOD profile. It has no value in relation to the profile. You do not need to clear unused input sections when creating profiles.
- You can enter any ASCII character, including spaces, with the exception of the comma. The comma is reserved to separate names in the other input sections of the dialog box

5. In the **LOD Access Strings** box, type names for LODs that you want to be accessible when using the profile.

Note:

Use a comma to separate each name.

6. (Optional) In the **Default LOD Labels** box, enter names for unlabeled LODs in older JT files.
7. Click **Add**.

The profile is created, and appears in the list of available profiles in the **3D Loader Preferences** dialog box. The input sections of the **Create New Profile** dialog box reset to let you create another profile.

8. (Optional) Create another profile by following the above steps.
9. Click **OK**.

Use an LOD Access Profile

Use LOD Access Profiles to control access to LODs present within .jt files.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **Use LOD Access Profiles when opening documents**.
3. In the **Profile** section, in **Profile**, select a profile.

The **Current Profile** header updates to match your selection.

4. Click **OK**.

Note:

- When opening files using a specific profile, only LODs labeled with the profile names are accessible during the work session. The profile filters out any other LODs present in the file.
- If you change LOD profiles during an active work session, the profile is in effect for files subsequently opened. The LOD content of the active file is not affected.

Modify an LOD Access Profile

Change the settings of an LOD Access Profile.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **Use LOD Access Profiles when opening documents**.
3. In the **Profile** section, in **Profile**, select the name of the profile that you want to edit, and click **Modify**.
4. In the **Modify Profile** dialog box, edit the appropriate profile information.
5. Click **OK**.

Delete an LOD Access Profile

Permanently remove an LOD Access Profile from your saved settings.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **Use LOD Access Profiles when opening documents**.
3. In the **Profile** section, in **Profile**, select the name of the profile that you want to edit, and click **Delete**.

The profile is deleted.

4. Click **OK**.

Reset an LOD Access Profile

Reset LOD Access Profile settings to their defaults.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **Use LOD Access Profiles when opening documents**.
3. In the **Profile** section, click **Reset All**.

A warning appears, noting that this option deletes all changes made to the LOD Access Profile settings. You can choose to cancel or continue.

4. Click **OK**.

The LOD Access Profile settings are reset to their defaults.

5. Click **OK**.

Enable LOD Access Profile warning messages

Specify to display warning messages if unused access strings are found in a profile, or if no part LODs match the profile.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, choose any of the following:

To	Do this
Display a message when unused access strings are found in a profile.	LODProfile Warning: Found unused access strings in profile Note: This message is displayed when access strings in a profile are not used to load a part. When this happens, the defective LOD and its label are removed from the part. The following situations may result in unused LOD labels in profiles: • Spelling mistakes in the part LOD label • Spelling mistakes in the access strings
Display a message when the use of a LOD profile results in no part LODs being accepted.	LODProfile Warning: Removed all LODs Note: This message is displayed when the use of an LOD profile results in no part LODs being loaded. When this happens, all LODs are removed from the part and the geometry cannot be displayed in the Viewing window. This may occur in the following situations: • The wrong profile is used for the part • A mismatch exists between default labels and access strings • Spelling mistakes in the part LOD label • Spelling mistakes in the access strings

3. Click **OK**.

Overview of LOD Access Profile metadata

When errors occur which lead to the display of LOD Access Profile warning messages, the problematic parts are tagged with metadata properties intended to assist you in analyzing the situation. The metadata is associated with parts as follows:

For	The parts are tagged with this metadata
Part problems that result in the display of the LODProfile Warning: Found unused access strings in profile warning message	Key: PROFILE_UNUSED_ACCESS_STRINGS Value: True
Part problems that result in the display of the LODProfile Warning: Removed all LODs warning message	Key: PROFILE_REMOVED_ALL_LODS Value: True

These metadata properties are intended to help you find the parts with problems. An example analysis scenario might include the following steps:

1. Disable LOD profiles.
2. Load the part.
3. Examine the LOD labels with the Part Edit functionality.
4. Make the necessary changes.

Specify NX PRT file data and tessellation parameters

A typical 3D PRT file may contain pre-tessellated facet data and boundary representation XT B-Rep data. Use the **NX Part Load Preferences** to determine which data to load first and what tessellation parameters to use.

Note:

The settings in the **NX Part Load Preferences** affect the loading performance, measurement precision, and rendering quality of the data in the viewer.

Note:

You can view NX 3D PRT files under the following conditions:

- To view 3D PRT files that consist of a top-level assembly with subassemblies, you must save all of the components that make up your model in the same directory or use a search_opt.def file to specify other directories.

If your sub-component part files are not in the same directory as the top-level assembly file, create a text file called "search_opt.def" and place it in the same directory as your top-level assembly. In the "search_opt.def" file, type the absolute directory paths where the sub-component part files are located, placing each path on a new line. When you open the PRT file, the application uses the directory path specified in this file to find and load the sub-component part files.

1. Choose **File**→**Preferences**→**3D Loader**.
2. In the **3D Loader Preferences** dialog box, select **NX Part Load Preferences**.
3. Choose any of the following:

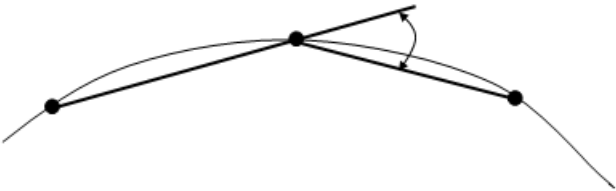
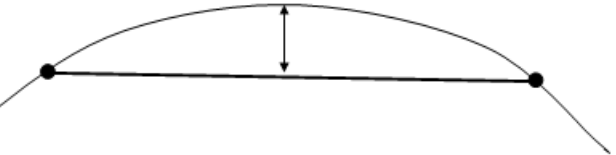
To	Select this
Load pre-tessellated facet data in the part file for display and measurement purposes, and ignore XT B-Rep data. If there is no facet data, XT B-Rep data will be used instead. Note: This setting may improve the loading performance based on the size of facet data. However, measurement is based on the facet data so it is not precise.	Use facet data, ignore B-Rep data
Load pre-tessellated facet data for display purposes, and late-load XT B-Rep data for precision measurement when necessary.	Use facet data for rendering, load B-Rep data on demand
Ignore facet data and use XT B-Rep data for display and measurement purposes. If there is no XT B-Rep data, facet data will be used instead. Note: This setting may slow down the loading performance because the application will create facets by tessellating the XT B-Rep data upon load.	Ignore facet data, use B-Rep data only

4. In the **Parameter values** area of the **Tessellation** section, choose **Absolute** or **Relative**.
 - Absolute: specify tessellation parameters in absolute units, such as document units or inches.
 - Relative: specify tessellation parameters relative to each part's size (bounding box diagonal length).

5. You can specify values for the following settings.

Tip:

For value settings, use positive decimals. Relative numbers are typically smaller than 1.0.

To	Select this
Determine the maximum angle between triangles along the boundary edges of the B-Rep. This is the angle formed by two consecutive line segments in the tessellation of a curve, as shown below. This parameter specifies the maximum angle allowed (smaller angles force finer tessellations). Note: The default value is 22.5 degrees. 	Maximum segment angle
Determine the maximum chordal length, which is the distance that the tessellation (straight line segment) has from the B-Rep curve. This parameter specifies the maximum deviation allowed (smaller values force finer tessellations). Note: The default value is 0.001. A line segment in the tessellation of a curve approximates the curve with a certain chordal deviation, as shown below. 	Maximum chordal deviation
Determine the maximum segment length, which is the length of the line segment in the tessellation of a curve, as shown below. This parameter specifies the maximum length allowed (smaller values force finer tessellations).	Maximum segment length

To	Select this
Note: The default value is 0.001. 	

- Click **OK**.

Change the load order for reference sets

You can specify the order in which the application attempts to load reference sets when a model is opened. The first reference set in the list that is found in the model is loaded.

Note:

The reference sets options in the **3D Loader Preferences** are applicable only to the standalone application viewer and the Lifecycle Viewer. To specify the reference set load order for embedded viewers within Teamcenter applications such as Structure Manager, you must use the **JT_FILE_PRIORITY_REFSETS** Teamcenter preference.

- Choose **File**→**Preferences**→**3D Loader**.
- In the **3D Loader Preferences** dialog box, click the **Reference Sets** tab.
- Do any of the following:

To	Do this
Change the reference set load order.	- In the Reference Set Load Order list, select the name of a reference set. - Click Move Up or Move Down.
Add a reference set to the load order.	- Click Add. - In the Add dialog box, type the name of the reference set. - Click OK.

To	Do this
Remove a reference set from the load order.	a. Select the name of a reference set. b. Click Remove. Note: The following default reference sets cannot be removed from the load order: • [Model] • [Entire Part] • [Empty] • [As Saved]
Load the default reference set according to the order specified in the Teamcenter JT_FILE_PRIORITY_REFSETS preference.	Select the Teamcenter Reference Set settings take precedence check box.

4. Click **OK**.

The next model you open that contains reference sets will load according to these settings.

Display the Active Layer Filter

You can display the Active Layer Filter when a model is loaded. If you turn on **Set Default Layer Filter**, the filter is applied without making the geometry visible. If you turn on **Show all components in the Default Layer Filter**, any default layer geometry is also made visible.

1. Choose **File→Preferences→3D Loader**.
2. On the Other page, select the **Set Default Layer Filter** check box.
3. (Optional) Select the **Show all components in the Default Layer Filter** check box.

When you load a model that contains an Active Layer Filter, it is displayed automatically.

Specify the default behavior for end items

You can specify for the application to override end items.

Note:

The **Show end item structure** setting is applicable only for structure saved locally as PLM XML and loaded in the standalone application viewer.

1. Choose **File→Preferences→3D Loader**.
2. In the **3D Loader Preferences** dialog box, click the **Other** tab.
3. In the **General** section, select the **Show end item structure** check box.

Display load failure error messages

You can specify to display an error message when 3D files fail to load.

1. Choose **File→Preferences→3D Loader**.
2. On the **Other** page of the **3D Loader Preferences** dialog box, select the **Show Load Failure Dialogs** check box.

Display all parts automatically

You can automatically display all of the parts of a model when it is opened.

1. Choose **File→Preferences→3D Loader**.
2. On the **Other** page of the **3D Loader Preferences** dialog box, select the **Show all parts on document open (Not Recommended)** check box.

Load geometry lazily

You can specify for geometry to load only until the **global memory budget** is met.

Note:

This option is on by default. If **Load geometry lazily** is turned off, the application attempts to load **the entire specified assembly or subassembly** immediately.

Tip:

Disabling this option may be useful if you experience loading pauses while exploring a large 3D model.

1. Choose **File→Preferences→3D Loader**.

- On the Other page of the **3D Loader Preferences** dialog box, select the **Load geometry lazily (Recommended)** check box.

Turn on background processing for JT files

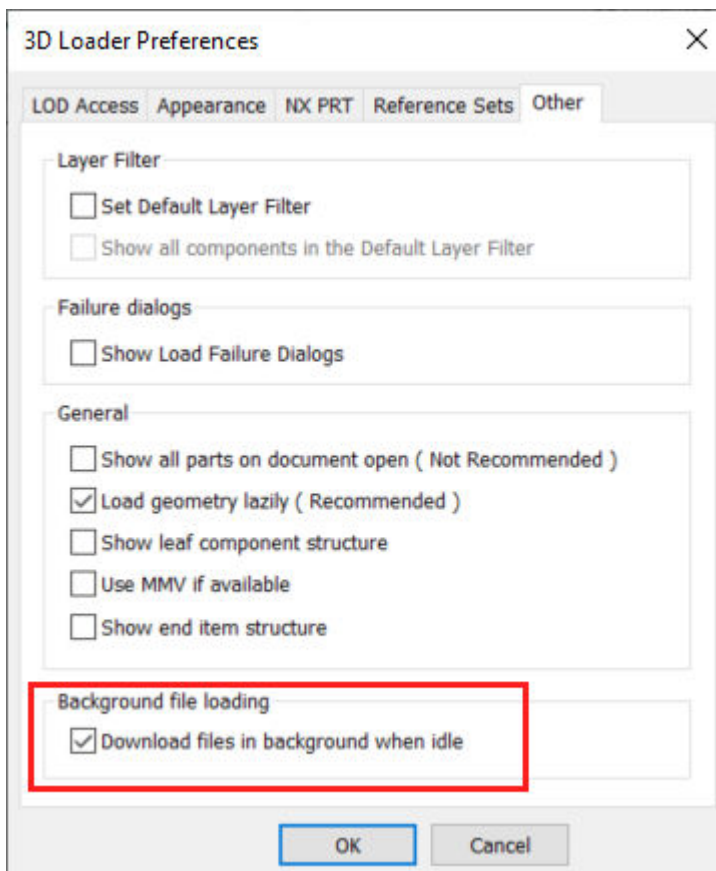
Using background file loading for large JT models may improve the performance of subsequent analysis operations, for example creating a cross section of a vehicle engine.

You can select background loading when your JT models are saved in a Teamcenter database, and when your environment includes a 9.1 or newer FCC client that is configured for background loading.

Note:

If you want to turn on background loading, do this before you load JT files.

- Choose **File→Preferences→3D Loader**.
- Navigate to the **Other** page in the **3D Loader Preferences** dialog box.
- In the **Background file loading** section, select **Download files in background when idle**.



Note:

- Your administrator installs and configures the proper FCC client, making sure to:
 - Install FCC 9.1 or later.
 - Review and set FCC configuration options for background loading.

For more details, see [Configure FCC for background file loading](#).

Automatically pack assembly elements

Packing groups identical elements in a single level of an assembly. When either the **Show structure packed** or **Use Teamcenter Preference for packing on expansion** preference is selected, Lifecycle Visualization will attempt to automatically pack any items that can be packed, based on the packing criteria settings defined in the following Teamcenter preferences.

- **BOMExcludeFromPackCheck** and **BOM_Additional_Packing_Criteria** — determine which items are considered "packable."
- **PSEAutoPackPref** — determines whether a structure is automatically packed within Teamcenter.

1. Choose **File** tab→**Preferences**→**3D Loader**.
2. Navigate to the **Other** page in the **3D Loader Preferences** dialog box.
3. In the **General** section, select one of the following.

Preference	Action
Show structure packed	Automatically packs the structure within Lifecycle Visualization and is not affected by the Teamcenter PSEAutoPackPref preference setting.
Use Teamcenter Preference for packing on expansion	Automatically packs the structure within Lifecycle Visualization based on the Teamcenter PSEAutoPackPref preference setting.

Navigating through models

3D navigation overview

You can view your 3D models from any direction and any point within the Viewing window. Navigating through 3D models entails changing the position and orientation of the navigation camera. For example, when you use the rotate command, the model may appear to spin in place but in actuality it is the navigation camera that moves.

You can set your navigation mode on the **3D Navigation** toolbar or from the **Navigation** menu.



3D Navigation toolbar

When navigating a model, you can:

- Examine your model by panning, zooming, and rotating the navigation camera. The model appears to follow your mouse movements around the screen.
- Center or align the navigation camera with a point in the Viewing window.
- Fly through your model by moving forward, back, left, right, up, and down. Your mouse movements control the movement of the camera as it flies through the model.
- Explore your model. The explore navigation mode constrains the camera movement so that you appear to walk through your model. The constraints allow you to move forward and back, and to look left, right, up, and down. Use the arrow keys or your mouse to move the camera through your model.
- Align the navigation camera with a preset viewing angle.
- Specify 3D navigation preferences. Change options such as the navigation style, the rate of acceleration, and the camera height when in terrain following mode.
- Specify 3D performance settings. Adjust a number of settings to improve navigation performance in the 3D Viewing window.
- Customize the preset performance options available on the **3D Performance** toolbar.
- Review guidelines that may help you to improve your navigation performance.
- Disable advanced viewing options to troubleshoot graphics performance problems.

Examine your model

You can move the eye point around your model to view it from any angle and any distance. The model follows your mouse movements around the screen.

1. Choose **Navigation→Mode→Examine**.
2. Navigate through your model in one of these ways:

To perform this task	On the 3D Navigation toolbar choose this option	Or hold this key as you move the mouse cursor
Return to the last static view	Previous View	No keyboard shortcut available
Move to the next static view	Next View	No keyboard shortcut available
Fit all visible parts into the Viewing window	Fit All 	No keyboard shortcut available
Zoom to a specific area of your model	Zoom Area 	No keyboard shortcut available
Center a point of your model. This point becomes the center of rotation	Seek 	No keyboard shortcut available
Pan your model in the plane of the Viewing window	Pan 	Ctrl
Zoom closer to or farther from your model	Navigate 	Shift
Rotate around your model using the mouse	Rotate 	Shift+Ctrl
Rotate continuously around your model	Automatic Motion 	No keyboard shortcut available
Align the model to the normal of the seek point	Align to face (Navigation Preferences)	No keyboard shortcut available

Return to the previous view

As you move your mouse cursor around the Viewing window, you navigate within the model. Once you stop moving the mouse, the pointer stops. When the pointer is no longer moving, the system logs that as a static view.

On the **3D Navigation** toolbar, click **Previous View** .

Note:

This option is available in all three navigation modes: Examine, Explore, and Fly.

Move to the next view

As you move your mouse cursor around the Viewing window, you navigate within the model. Once you stop moving the mouse, the pointer stops. When the pointer is no longer moving, the system logs that as a static view.

On the **3D Navigation** toolbar, click **Next View** .

Note:

This option is available in all three navigation modes: Examine, Explore, and Fly.

Fit all visible parts into the Viewing window

Fit all the parts that are currently displayed into the Viewing window.

- On the **3D Navigation** toolbar, click **Fit All** .

Zoom to an area

Zoom to a specific area of your 3D model, which you define in the Viewing window.

1. On the **3D Navigation** toolbar, click **Zoom Area** .
2. Drag the mouse cursor across the model in the Viewing window.

The view zooms to the model area that you have specified.

Note:

When you zoom a 3D model, you specify two of the dimensions when you drag the mouse cursor in the Viewing window. The third dimension is the lookat point of the model. You can change the lookat point of your model by choosing **Seek**.

Center a point of the model

When you center a point of your model in the Viewing window, the point becomes the center of rotation.

1. If necessary, set zoom **preferences** for **Seek**.
2. Do any of the following:
 - On the **3D Navigation** toolbar, click **Seek** , and then click a point on the model.
 - (If you are navigating using Examine mode) Click a point on the model using the middle mouse button (or wheel).
 - (If you are navigating using Examine mode) Right-click a point on the model and choose **Seek**.

Your view changes so that the point you clicked is at the center of the Viewing window.

Align the model view to the normal of a face

When navigating through your model using Examine or Fly modes, you can align a specified point on the model to the normal of a face.

1. On the **3D Navigation** toolbar, click **Preferences** .
2. In the **Preferences** dialog box, click the **Seek** tab, make sure the **Align to Face** check box is selected, and click **OK**.
3. Do any of the following:
 - On the **3D Navigation** toolbar, click **Seek** , and then click the face.
 - (If you are navigating using Examine mode) Point to the face and click the middle mouse button (or wheel).
 - (If you are navigating using Examine mode) Right-click the face and choose **Seek**.

Your view changes so that the point you clicked is at the center of the Viewing window and your viewing angle is aligned with the normal of the surface at that point.

Rotate around the model

1. (Optional) Change the rotation **preferences**.
2. On the **3D Navigation** toolbar, click **Rotate** .
3. Drag the cursor around the Viewing window.

Your eye point rotates around the model in the Viewing window.

Tip:

You can open the **Rotate** dialog box to rotate the view a specified degree amount. You can also rotate the view by using one of the options available on the **Standard Views** toolbar.

Rotate around a point in the model

Select a point on the model as the point of rotation. After you select the point of rotation, moving the mouse rotates the model around the selected point.

1. Place the cursor over the point on the model you want to set the rotation point.
2. Do one of the following:
 - Press and hold the left mouse button without moving the cursor until the rotation sphere appears beneath your cursor.
 - Right-click and choose **Rotate around a user defined point** from the dropdown menu. A rotation sphere appears.



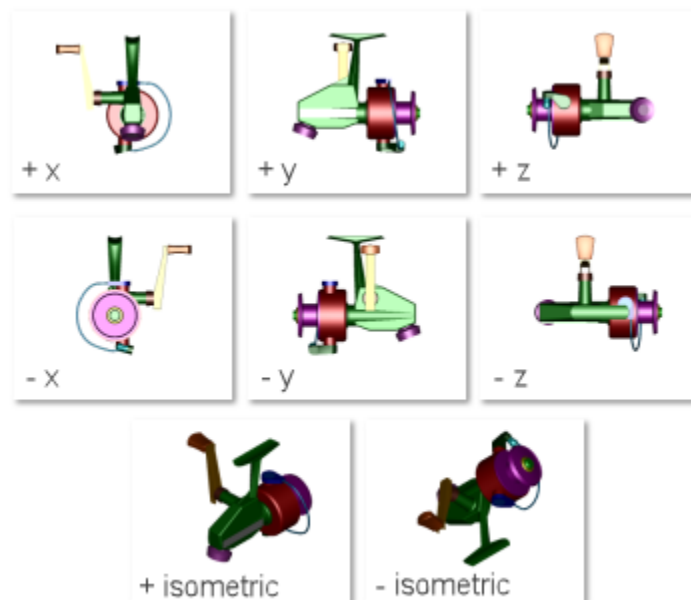
If you click away from the model and then select **Rotate around a user defined point** again, the model will rotate around the last selected point.

3. Once the rotation sphere appears, drag the cursor to rotate around that point.

When you release the mouse the rotation sphere is removed and navigation returns to rotate around the center of the viewer.

Rotate your view to align to an axis

Quickly switch between a number of predefined 3D viewing angles.



1. On the **3D Standard Views** toolbar, click **Align to Axis** .
2. Choose one of the following:

To align your view to the	Choose
X-axis looking in the negative X direction	-X
Y-axis looking in the negative Y direction	-Y
Z-axis looking in the negative Z direction	-Z
X-axis looking in the positive X direction	+X
Y-axis looking in the positive Y direction	+Y
Z-axis looking in the positive Z direction	+Z
Isometric view looking in the positive XYZ direction	Isometric
Isometric view looking in the negative XYZ direction	-Isometric

Tip:

You can also align your view to an axis from the **Rotate** dialog box.

This option is available at the Professional service level and above.



1. (If the **Rotate** dialog box is not already open) On the **3D Standard Views** toolbar, click **Enable View Control** .
2. In the **Rotate** dialog box, click **+X**, **+Y**, **+Z**, **-X**, **-Y**, or **-Z**.

Rotate your view a specified number of degrees

1. On the **3D Standard Views** toolbar, click **Enable View Control** .

The **Rotate** dialog box appears. The default rotation angle is 90 degrees.



2. (Optional) Type the number of degrees you want to rotate the view. You can type any value between -360 and 360.

3. In the **Rotate** dialog box or on the **3D Standard Views** toolbar, choose one of the following:

- **Rotate Clockwise** 
- **Rotate Up** 
- **Rotate Counter-clockwise** 
- **Rotate Left** 
- **Rotate Down** 
- **Rotate Right** 

Rotate 3D models continuously

You can continuously rotate the navigation camera around the model, creating a spinning turntable effect.

1. If you are not in the Examine navigation mode, on the **3D Navigation** toolbar, click **Examine** .
2. Click **Preferences** .
3. In the **Examine Navigation Preferences** dialog box, from the **Automatic Motion** list, choose **Rotate around Model**.
4. Adjust the following settings to control the behavior of the automatic rotation:
 - If you want to ensure that your models are displayed in their entirety within the 3D Viewing window, select the **Perform Fit All** check box.
 - If you want to change the direction of the model rotation, from the **Immediately Rotate** list, choose **Left** or **Right**.
 - If you want to change the speed of the model rotation, click the **Degrees / Second** slider and drag it left (slower rotation) or right (faster rotation). You can also type a value from 0 to 360, where 0 equals no rotation and 360 equals the fastest possible rotation.
5. Click **OK** to close the navigation preferences.
6. On the **3D Navigation** toolbar, click **Automatic Motion** .

The model begins to continuously rotate.

Tip:

You may need to click anywhere in the Viewing window to begin the model rotation.

7. To stop the rotation, click **Automatic Motion**  again.

Pan your model

Pan your model in the plane of the Viewing window.

1. On the **3D Navigation** toolbar, click **Pan** .
2. Press the left mouse button and move the pointer around the Viewing window.

The model appears to move with the mouse cursor.

Zoom around 3D models

Zoom closer to or farther from your 3D model in the Viewing window.

1. On the **3D Navigation** toolbar, click **Navigate** .
2. Press the left mouse button and move the cursor up or down in the Viewing window.

Your eye point moves closer to or farther from the model in the Viewing window.

Fly through your model

Flying navigation mode constrains your movement so that it appears you are flying through your model. Using constraints you can move forward and back, turn left and right, and look up and down. You can also move inside parts.

While flying through your model, control your direction by rotating or moving forward and backward. You can also increase and decrease your speed incrementally.

1. On the **3D Navigation** toolbar, click **Fly** .
2. Navigate through your model in one of these ways:

On the 3D Navigation toolbar, or from the Navigation menu choose this option	To perform this task
Previous View	Return to the last static view state.
Next View	Move view to the next static view.
Fit All	Fit all visible parts into the Viewing window.
Zoom Area	Zoom to a specific area of your model.
Seek	Center a point of your model. This point becomes the center of rotation. You can also align the centered model with the normal of the surface at that point.
Pan	Pan your model in the plane of the Viewing window.
Level	Return your view of the model to level .
Terrain Following (toolbar) Follow Terrain (menu)	Remain a specified distance above the "terrain" upon which you are moving.
Collision Detection (toolbar) Stop on Collision (menu)	Stop moving when you encounter an obstacle.
Lock up vector (Navigation Preferences)	Force your model to remain upright as you examine it .

Fly forward, back, left, or right through your model in the Viewing window.

To	Hold the left mouse button as you
Change the fly direction – fly forward	Press the W key and hold the mouse cursor near the crosshairs that indicate the position of the initial mouse click.
Change the fly direction – fly backward	Press the S key and hold the mouse cursor near the crosshairs that indicate the position of the initial mouse click.
Decrease speed by 10%	Press the A key.
Increase speed by 10%	Press the D key.

Tip:

You can press Ctrl+N to change navigation modes.

Look around your model

Look around as you fly through your model.

1. On the **3D Navigation** toolbar, click **Rotate in Place**.
2. Press and drag the left mouse button.

Your view follows the mouse cursor around the model. To move your view faster, move the pointer farther from the point at which you clicked; to move slower, move the pointer closer to the point at which you clicked.

Return the view of your 3D model to level

Return the view of your 3D model to level as you move through your model. The way it behaves depends on how the **Up Axis** and **Level** preferences are set.

Note:

When you select Fly navigation mode, the current view of the model is defined as level.

1. (Optional) Change the **Up Axis** and **Level preferences** of the model, if necessary.
2. On the **3D Navigation** toolbar, click **Level** .

The view in the Viewing window returns to level.

Follow the terrain

Follow the terrain as you move through your 3D model.

1. (Optional) Set terrain following **preferences**.
2. On the **3D Navigation** toolbar, click **Terrain Following** .
3. Move through your model.

You move no lower than the specified distance above the floor of your model.

Detect collisions

Use collision detection as you move through your 3D model to stay within a specified distance from an object.

1. (Optional) Set the navigation **preferences**.
2. On the **3D Navigation** toolbar, click **Collision Detection** .

When you move through your model, you stop when you are within the **Minimum Distance Allowed** of an object.

Explore your model

Explore navigation mode constrains your movement so that it appears you are walking through your model. Using constraints you can move forward and back, and look left, right, up, and down. Use the keyboard keys – W, A, S, or D keys or your mouse to move through your model.

Tip:

You can explore very large models, such as buildings or vehicles. Use **Examine** mode and your mouse to place yourself inside the model before you start **Explore** mode.

1. On the **3D Navigation** toolbar, click **Explore** .
2. Navigate through your model in one of these ways:

On the 3D Navigation toolbar, or from the Navigation menu, choose this option	To perform this task
Previous View	Return to the last static view state.
Next View	Move view to the next static view.
Fit All	Fit all visible parts into the Viewing window.
Zoom Area	Zoom to a specific area of your model.
Seek	Look at a point on your model.
Position	Choose your initial position in the model.
Pan	Pan your model in the plane of the Viewing window.
Level	Return your view of the model to level .
Terrain Following	Remain a specified distance above the "terrain" upon which you are moving.
Collision Detection	Stop moving when you are within a specified distance of an object.
Go Home	Return to the location and position you were at when you last selected Set Home .
Set Home	Set the current location and position as the home location.

You can use the following keys to navigate **Explore** mode:

To do this	Move the pointer	Or press this key
Move forward	Up in the Viewing window	W
Move backward	Down in the Viewing window	S
Turn left	Left in the Viewing window	A
Turn right	Right in the Viewing window	D
Look up	Not applicable	Shift+W
Look down	Not applicable	Shift+S
Look to the left	Not applicable	Shift+A
Look to the right	Not applicable	Shift+D
Increase the rotation factor	Not applicable	R
Decrease the rotation factor	Not applicable	F
Increase the translation factor	Not applicable	T
Decrease the translation factor	Not applicable	G

Tip:

You can press Ctrl+N to change navigation modes.

Look at an object on your model while exploring

While in Explore mode, pick an object on your model that you want to look at.

1. From the **Navigation** toolbar, choose **Seek**.
2. Click an object on the model.

The Viewing window changes by centering the view on the selected object.

Choose your initial position in the model

While in Explore mode, place your initial viewing position inside your model by choosing your position.

1. On the **3D Navigation** toolbar, click **Position** .
2. Click a point on your model.

The view in the Viewing window changes. The view appears as if you are standing on the floor of your model at the point you clicked.

Go to a set location and position

Return to the location and position you set as your home position.

1. **Set** a home position.
2. Move around in your model.
3. From the **Navigation** toolbar, choose **Go Home** .

The view changes to the location and position that was set as the home position.

Set a home location and position

Designate a location and position as home.

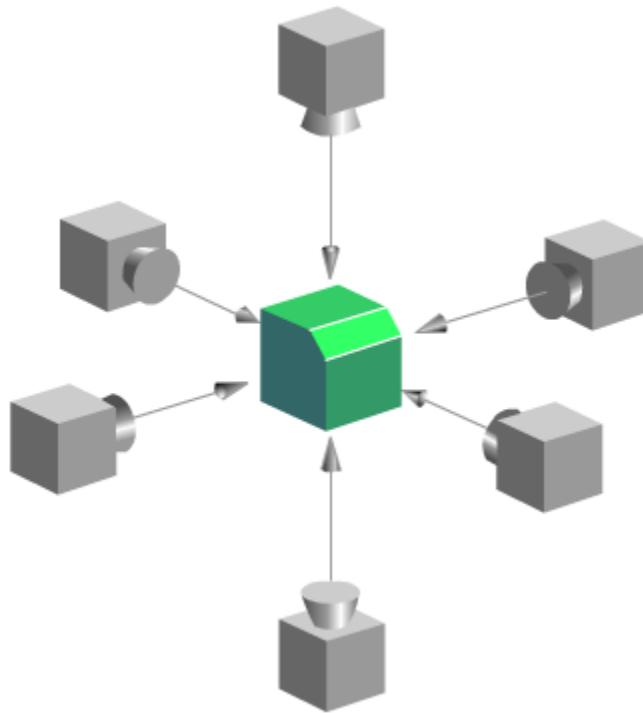
1. Navigate in your model to the location and position you want to save.
2. From the **Navigation** toolbar, choose **Set Home** .

The position is saved as the home position. This will remain the home position until another location and position are saved as the home position.

Standard views

Introduction to standard views

Standard views are defined by the camera Orientation, Position, and Projection (orthographic or perspective). You can use them to view any model. Out of the box, there are eight defined standard views.



You can:

- Create new standard views.
- Edit existing views.
- Save your standard views to a file and share them with others.
- Specify a standard view as the default view to use when you open any model.

View your model from a standard view

You can view any model from predefined standard views.

- Do any of the following:
 - Right-click in the background of the Viewing window, choose **Standard Views**, and then select the name of the view.
 - On the **3D Standard Views** toolbar, click **Edit Views** , and then double-click the name of the view.

Note:

There is a navigation preference to lock the camera up vector as you rotate your view. When you set **Lock up vector** to an axis, you cannot select axis-aligned standard views whose view vectors coincide with the locked up vector.

Specify a default view to use when opening all models

You can specify a standard view to use by default when you open any model.

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. (Optional) **Create** a new standard view.
3. In the **Standard Views** dialog box, from the **View List**, select the view and click **Set as default view**.

Tip:

If you have not set a default view, the system uses a default view that is not on the View List. If you set a default view and then want to return to the original system default, create a new view with the following parameters:

In this section	Specify
Orientation	Angles with X, Y, and Z set to 0
Position	Auto-set
Camera	Projection as Perspective

You can then click **Add**, give it a name, select it in the list, and click **Set as default view**.

Creating and editing standard views

Overview of creating and editing standard views

Standard views can be used with any model. They give you a way to view any model from one of several consistent views.

Out of the box, there are eight predefined standard views. You can create new views and edit existing views. You can also export your standard views to a file and share the file with other users.

Standard views have a camera Orientation, Position, and Projection mode.

- **Orientation** is your viewing direction relative to your **Look from** point.

You can change orientation or viewing direction by changing X, Y, and Z rotation angles. The X,Y,Z angle of 0,0,0 result in the top view, or a view from a point on the positive Z axis. The angle 0,0,30 results in a top view that has been rotated 30 degrees about the Z axis.

You can also change Orientation by specifying the **Look from** or **Look at** points, and the **Up vector**.

- **Position** can be **Auto-set** or a specific **Look from** point.

When position is **Auto-set**, the **Look at** and **Look from** points are only used to determine the viewing direction. Magnification will not change. If you specify a **Look from** point, magnification is determined by the distance between your **Look from** and **Look at** points.

- **Projection** is either **Perspective** or **Orthographic**. (The eight predefined standard views use **Orthographic** projection.)

Tip:

To see how Orientation and Position values control your view, leave the **Standard View** dialog box open as you rotate, pan, and zoom in the Viewing window. Watch the values change as you navigate around the model.

Make the current view a Standard View

1. In the Viewing window, rotate, zoom, and pan to the desired view.
2. On the **3D Standard Views** toolbar, click **Edit Views** .
3. (Optional) Set the **Camera Projection** to **orthogonal or perspective**.
4. In the **Standard Views** dialog box, click **Add**.
5. In the **View Name** dialog box, enter a name for the view.
6. Click **OK**.

The new view is added to the **View List**.

7. (Optional) Use the arrow keys in the to reorder the **View List**.

Note:

Your new view is automatically saved locally, once you add it to the list. You can also **save** the entire view list to a file to share with others.

Create a standard view by specifying parameters

Standard views are defined by the camera Orientation, Position, and Projection (orthographic or perspective). You can define a new standard view by specifying the Orientation, Position, and Projection.

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. In the **Orientation** section, do any of the following to specify the camera orientation:

To specify	Do this
Specify angles	Click Angles , and type values for X , Y , or Z .
Enter the coordinates of the Look at point	a. Click Look at and type the coordinates of the point. b. Type the Up vector coordinates or choose the Up vector axis from the list.
Pick the Look at point in the Viewing window	a. Click Look at. b. From the Part/Point list, choose Part or Point. c. Pick a part in the Viewing window or a point on the model. d. Type the Up vector coordinates or choose the Up vector axis from the list.

3. In the **Position** section, do any of the following to specify the camera position:

To	Do this
Place the camera along the viewing line	Click Auto-set .
Enter the coordinates of the Look from point	Click Look from and type the coordinates of the point.
Pick the Look from point in the Viewing window	a. Click Look from. b. From the Part/Point list, choose Part or Point. c. Pick a part in the Viewing window or a point on the model.

4. In the **Camera Projection** section, click **Orthographic** or **Perspective**.
5. (Optional if you use Perspective) Click the **Field of View** check box and type a new value.

6. (Optional) Click **Apply view** to preview the changes in the Viewing window.
7. In the **Standard Views** dialog box, click **Add**.
8. In the **View Name** dialog box, type a name for the view and click **OK**.

The new view is added to the **View List**.

9. (Optional) You can use the arrow keys in the **Standard Views** dialog box to reorder the views in the **View List**.

Note:

New views are automatically saved locally, once you add it to them list. You can also **save** the view list to a file to share with others.

Save your standard views to a file

You can save your standard views list to a file and share it with others.

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. In the **Standard Views** dialog box, click **Save**.
3. In the **Save As** dialog box, browse to the appropriate file path and enter a file name.
4. In **Save as Type**, choose **View Files (*.vfv)** or **Reg Files (*.reg)**.
5. Click **Save**.

Load a list of standard views

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. In the **Standard Views** dialog box, click **Load**.
3. In the **Open** dialog box, select the .reg or .vfv file to open.
4. Click **Load**.

The names of the standard views saved in the file appear in the **View List**.

Edit a standard view

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. Select the view to edit in from the **View List** in the **Standard Views** dialog box.
3. Do any of the following:
 - In the Viewing window, rotate, zoom, or pan to change the view.
 - In the **Standard Views** dialog box, edit view parameters and click **Apply view** to see the changes.
4. Click **Replace** to save the changes.

Rename an existing standard view

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. In the **Standard Views** dialog box, in the **View List** section, select the view you wish to rename.
3. Click **Rename**.
4. In the **View Name** dialog box, type the new name of the view and click **OK**.

Delete a standard view

1. On the **3D Standard Views** toolbar, click **Edit Views** .
2. In the **Standard Views** dialog box, select one or more views from the **View List**, then click **Delete**.

The view is deleted from the **View List**.

Use multi-touch gestures to view 3D models

Teamcenter lifecycle visualization supports multi-touch gestures on gesture-enabled hardware.

- To perform a left mouse click for selection, tap the screen with a single finger and release.
- To perform a left mouse click for camera rotation, tap the screen with a single finger and drag.
- To perform a right mouse click to bring up the context menu, tap the screen with a single finger and hold.

- To zoom the camera in or out, tap the screen with two fingers and move the fingers apart or toward each other.
- To rotate the camera, tap the screen with two fingers and move the fingers in opposing directions. You can also pivot one finger around the other.
- To pan the camera, tap the screen with two fingers, where the target is the midpoint between the fingers.

Set examine navigation preferences

1. On the **3D Navigation** toolbar, click **Examine** .
2. On the **3D Navigation** toolbar, click **Preferences** .
3. In the **Examine Navigation Preferences** dialog box, adjust any of the following settings:

General tab	
In this section	Do this
Navigation Control	Specify a navigation Style to use. a. Choose one of the following: • Visualization (Classic): When rotating, if you click and drag the cursor near an outer edge of the Viewing window, the same axis of rotation is used, regardless of which edge the cursor is near. • Visualization: When rotating, if you click and drag the cursor near an outer edge of the Viewing window, different axes of rotation are used. If the cursor is near the left or right edge of the Viewing window when you begin, a horizontal axis is used for rotation. If the cursor is near the bottom edge, a vertical axis is used; if the cursor is near the top edge, a perpendicular axis is used. • CAD1 (NX): Provides a general navigation capability similar to that of NX CAD products. Use the mouse buttons to do the following: • Drag the middle button to rotate. • Drag middle + left to zoom. • Drag middle + right to pan. • Drag the left button to Select Parts by Area.

	Note: If you choose Rotate, Pan, or Navigate options, you temporarily switch to Visualization style. Choose the same option again to return to CAD1 (NX) style. • CAD2: Provides a general navigation capability similar to that of other major CAD products. Use the mouse buttons to do the following: • Drag the middle button to pan. • Drag middle + left or right to rotate. • Hold middle, click left or right, and drag to zoom. • Drag left button to Select Parts by Area. Note: If you choose Rotate, Pan, or Navigate options, you temporarily switch to Visualization style. Choose the same option again to return to CAD2 style. b. Select the Update style during session load check box to automatically set the navigation control style stored in the session file. c. Select the Flip mouse direction for zoom check box to specify how you use the mouse to zoom in the Viewing window. Note: The default behavior for Zoom  is to zoom out by dragging <i>up</i> and zoom in by dragging <i>down</i>. You can reverse this behavior by selecting this option. The option also reverses the direction you roll the mouse wheel to zoom in or out. This option is set for viewing all models and images. That is, if you specify the default behavior or the reverse behavior for zooming, the option is automatically set for viewing all 2D, 3D, and ECAD data.
View Transitions	Set the amount of time it takes for the scene in the Viewing window to change. a. Select the Transition time check box. b. Drag the slider to adjust the transition time, or type a value in the box.

	Example: If you select Transition time and specify a value of 10, the scene takes 10 seconds to interpolate from the current view to the Fit All view.
Automatic Motion	Change the rate at which you accelerate as you move through the model. - From the Automatic Motion list, choose Control Speed with Mouse and drag the Multiplier slider to the left to accelerate more slowly, or to the right to accelerate more quickly. - Drag the Multiplier slider to adjust the rate of acceleration, or type a value in the box.
Automatic Motion	Continuously rotate around the model when Automatic Motion is turned on. - From the Automatic Motion list, choose Rotate around Model. - To fit all visible parts in the Viewing window, select the Perform Fit All check box. - To adjust the direction of the rotation, select the Immediately Rotate check box, and then choose one of the following: Left Right - To adjust the rotation speed, drag the Degrees / Second slider or type a value in the box.
Rotation	Specify options to control the camera rotation when in standard rotation mode. - To prevent the camera from rotating more than 90 degrees from upright, do the following: - Select the Lock up vector check box. - From the rotation axis list, specify the up axis. Note: If you lock the up vector to an axis, you cannot select axis-aligned Standard Views whose view vectors coincide with the locked up vector. - To position the rotate point in the center of the Viewing window when you rotate the model, select the Center rotate point check box.
Display tab	
Crosshairs	Display crosshairs as you navigate in the Viewing window.

	- To display or hide the crosshairs at the point of your mouse click, select the Draw crosshairs check box. - To change the color of the crosshairs, select the color box, and then choose a color. To change the width of the lines in the crosshairs, from the Line Width list, choose a value between 1 and 5.
Seek tab	
Seek	Adjust options for Seek mode. - To adjust how much the camera zooms when you seek a point, select the Zoom percentage check box, and then drag the slider or type a value in the box. - To automatically align the model to the normal of a face, select the Align to the face check box.

- (Optional) To set the preferences on a page as default, on the page, select the **Set as Default** check box.
- Click **Apply** to apply the preferences without closing the **Navigation Preferences** dialog box, or click **OK** to apply the changes and close the dialog box.

Set fly navigation preferences

- On the **3D Navigation** toolbar, click **Fly** .
- On the **3D Navigation** toolbar, click **Preferences** .
- In the **Fly Navigation Preferences** dialog box, specify preferences for the appropriate options.

General page	
In this section	Do this
View Transitions	Set the amount of time it takes for the scene in the Viewing window to change. - Select the Transition time check box. - Drag the slider to adjust the transition time, or type a value in the box. Example: If you select Transition time and specify a value of 10, the scene takes 10 seconds to interpolate from the current view to the Fit All view.
Accelerated Motion	Change the rate at which you accelerate as you move through the model.

	Drag the Multiplier slider to adjust the rate of acceleration, or type a value in the box.
Up Axis	Prevent the model from rotating more than 90 degrees from upright. - Select the Lock up vector check box. - Select an axis to define as up.
Level	Specify level options. - Select Roll only to level the X- axis. - Select Pitch only to level the Y-axis. - Select Both to level both the X- axis and the Y-axis.
Display page	
Crosshairs	Display crosshairs as you navigate in the Viewing window. - To display or hide the crosshairs at the point of your mouse click, select the Draw crosshairs check box. - To change the color of the crosshairs, select the color box, and then choose a color. To change the width of the lines in the crosshairs, from the Line Width list, choose a value between 1 and 5.
Seek page	
Seek	Adjust options for Seek mode. - To adjust how much the camera zooms when you seeks a point, select the Zoom percentage check box, and then drag the slider or type a value in the box. - To automatically align the model to the normal of a face, select the Align to the face check box.
Intersect page	
Units	Change the default settings for measurement units. - For Model units, specify the units of measurement in which the model was created, if those units are not read from the model file. - For Dialog units, specify the units in which measurements are displayed in the dialog box. Note: If the Dialog Units you specify are different from the Model Units, the program converts the units automatically.

Terrain Following	Specify the minimum height above the terrain for terrain following. • Drag the Minimum height above terrain slider, or type a value in the box. • Select the Follow terrain normals check box to look up as you follow an incline, and look down as you follow a decline.
Collision Detection	Specify the minimum distance to define a collision. • Drag the Minimum distance allowed slider, or type a value in the box. • Select the Collision detect when moving backwards check box to detect collisions when moving in both directions. Clear the check box to detect collisions only when moving forward.

4. (Optional) To set the preferences on a page as default, on the page, select **Set as Default**.
5. Click **Apply** to apply the preferences without closing the **Navigation Preferences** dialog box, or click **OK** to apply the changes and close the dialog box.

Set explore navigation preferences

1. On the **3D Navigation** toolbar, click **Explore** .
2. On the **3D Navigation** toolbar, click **Preferences** .
3. In the **Explore Navigation Preferences** dialog box, specify preferences for the appropriate options.

In this section	Do this
Units	Change the default settings for measurement units. • For Model units, specify the units of measurement in which the model was created, if those units are not read from the model file. • For Dialog units, specify the units in which measurements are displayed in the dialog box. Note: If the Dialog Units you specify are different from the Model Units, the program converts the units automatically.
Terrain Following	Specify the minimum height above the terrain for terrain following. Drag the Height above terrain slider, or type a value in the box.
Collision Detection	Specify the minimum distance to define a collision. Drag the Minimum distance slider, or type a value in the box.

In this section	Do this
Up Vector	Define the up vector and constrain the model rotation to less than 90 degrees. From the Up Vector list, specify an axis.
View Transitions	Set the amount of time it takes for the scene in the Viewing window to change. - Select the Transition time check box. - Drag the slider to adjust the transition time, or type a value in the box. Example: If you select Transition time and specify a value of 10, the scene takes 10 seconds to interpolate from the current view to the Fit All view.
Sensitivity	Specify the percentage of the current speed that you want to move when you translate or rotate. - To adjust the percentage for translation, adjust the Translation slider, or type a value in the box. - To adjust the percentage for rotation, adjust the Rotation slider, or type a value in the box.
Crosshairs	Display crosshairs as you navigate in the Viewing window. - To display or hide the crosshairs at the point of your mouse click, select the Draw crosshairs check box. - To change the color of the crosshairs, select the color box, and then choose a color. To change the width of the lines in the crosshairs, from the Line Width list, choose a value between 1 and 5.

- (Optional) To set the preferences on a page as default, on the page, select **Set as Default**.
- Click **Apply** to apply the preferences without closing the **Navigation Preferences** dialog box, or click **OK** to apply the changes and close the dialog box.

Set 3D performance preferences

You can customize your settings to control the appearance of your model in the Viewing window.

- Do any of the following:
 - In the Viewing window, right-click and choose **Performance→Edit**.

- On the **3D Performance** toolbar, click **Edit Performance Settings** .

The **Performance** dialog box has six tabs, **Rendering**, **Size Culling**, **Distance Culling**, **Toolbar**, **Global**, and **Troubleshoot**.

- On the **Rendering** tab, change any of the following options:

Use this option	To do this
Image Quality (Moving Frames and Still Frame)	Adjust the level of detail displayed on your model in the Viewing window. Move the Image Quality slider to the right to increase the level of detail or to the left to decrease the level of detail. Tip: To show more details, move the Image Quality slider more towards high. To increase interactivity, move the Image Quality slider more towards low.
Advanced Transparency (Moving Frames and Still Frame)	Improve the quality of transparent parts. Transparent parts can be rendered without regard for the parts' geometry. This may lead to a performance increase in the Viewing window, but it may also lead to visual artifacts. Using Advanced Transparency ensures that the visual accuracy of transparent parts is maintained. Note: When viewing a model that contains multiple transparent parts that overlap one another, visual artifacts may appear even if the Advanced Transparency is enabled.
Cast shadows defined by light sources (Moving Frames and Still Frame)	Enable or disable shadow generation for the current active view.
Fixed Frame Rate (Moving Frames and Still Frame)	Control the rate at which your model is redrawn in the Viewing window. Move the Fixed Frame Rate slider to the left to decrease the number of frames per second that are drawn while the model is moving, or move the slider to the right to increase the number of frames per second that are drawn while the model is moving.

Use this option	To do this
	Tip: Enter values for low and high to control the range of the Fixed Frame Rate slider.
Show bounding boxes for parts that are not yet loaded; type ()	Display the bounding box of a part that is not yet visible due to background loading. You can specify how the bounding box is displayed by choosing one of the following options: • Wireframe • Solid • Automatic Note: The Automatic bounding box option determines how well a part fills its bounding box, and automatically displays the bounding box as either a solid or a wireframe.

3. On the **Size Culling** tab, change any of the following options:

Use this option	To do this
Cull parts with less than () % estimated screen coverage (Moving Frames and Still Frame)	Specify the percentage of the Viewing window that a part must cover before the part is drawn. Parts that are smaller than this size are not drawn in the Viewing window. To set the size parameter, you can select one or more parts, and click Set from selected parts . If you select several parts, the value is set to the maximum of all parts' values.
Cull parts with a bounding box diagonal less than () mm (Moving Frames and Still Frame)	Specify the minimum part size to display. Parts are displayed only if the diagonal measurement of the bounding box of the part is larger than the size value. To set the size parameter, you can select one or more parts, and click Set from selected parts . If you select several parts, the value is set to the maximum of all parts' values.

4. On the **Distance Culling** tab, change any of the following options:

Use this option	To do this
Cull parts beyond () mm (Moving Frames and Still Frame)	Specify the number of millimeters that a part must cover before being drawn. Parts that are smaller than this size are not drawn in the Viewing window. In order to set the

Use this option	To do this
	size parameter, you can select one or more parts, and click Set from selected parts . If several parts were selected, the value is set to the maximum of all parts' values.
Fog	Show fog on your model in the Viewing window. Note: - The color of all drawn geometry is mixed with the fog color. The geometry closest to you maintains more of its original color, and geometry further from you assumes more of the fog color. - Geometry beyond a certain distance fades completely into the fog color and becomes invisible. This effect automatically sets a distance culling value (independent of the distance culling you set) and improves drawing performance. - Fog affects both moving and still frames in the same manner. - When you set the fog color, the background color specified in the 3D Viewing Preferences dialog box is temporarily overridden.
Color	Select the fog color from the list.
At distance	Set the distance for fog calibration. The fog appears at the specified distance. - Select one or more parts on the model. - Click Set from selected parts. The distance for fog calibration is set. If you selected more than one part, the distance value is set to the minimum of all parts' values.
Density	Specify how dense you want the fog to appear at the specified distance. Type a number between 0.0 and 1.0, or use the slider. Note: - A value of 0.0 results in no fog. - A value of 1.0 results in the geometry at the given distance being drawn entirely in fog. Parts that fall into the range of a 1.0 density are not visible in the Viewing window.

5. On the **Toolbar** tab, change any of the following options:

Use this option	To do this
New	Create a new performance button.
Rename	Rename a performance button.

Use this option	To do this
Delete	Delete a performance button.
Move Up, Move Down	Change the order of the 3D performance buttons on the toolbar.
Update from 3D window	Store the performance settings of the current 3D window into the selected performance setting button.
Apply to 3D window	Apply the performance settings of the selected performance setting button to the current 3D window.

6. On the **Global** tab, change any of the following options:

Use this option	To do this
Limit: () % of available memory	Displaying very large model files typically requires large amounts of memory. To prevent the application from using memory that your system may need for other programs to remain responsive, you can specify the maximum percentage of RAM available for displaying 3D models. If model files exceed the memory limit, further part loading is halted and parts not rendered in the last few frames are unloaded until memory usage falls below the specified percentage. It is not recommended, but you also can force the application to use any virtual memory available on your machine, including all of the paging file. When memory consumption exceeds the amount of physical RAM, application responsiveness decreases drastically. To use virtual memory, type a number greater than 100. For example, if you want to use 50 percent of your virtual memory, type <i>150</i>. Tip: Current memory usage within the application is displayed on the right side of the status bar in the Mem section. The first number displayed is the actual memory in use; the second number is your specified memory budget.
Shadow edge softness	Soften shadow edges to give lighting a more natural appearance. Choose from the following settings: • None • 4 Sample • 16 Sample

Use this option	To do this
	Note: - The visual quality of shadow edges increases with the number of samples used. The default setting is None. - Shadow edge softness settings are available only on platforms that support programmable shaders. - Enabling shadow edge softness can have a significant negative impact on rendering performance. It is most useful when set to a low shadow map resolution, such as Course or Medium. The beneficial effects of edge softness are less pronounced at higher shadow map resolutions.
Shadow map resolution	Specify shadow map resolution. Choose from the following settings: Coarse Medium Fine Extra Note: The visual quality of shadow edges increases when higher resolution shadow maps are used. The default setting is Course, which is the lowest resolution available.

7. On the **Troubleshoot** tab, change any of the following options:

Use this option	To do this
OpenGL slider	Update the effective OpenGL level used by the application. Do one of the following: - Move the slider to the left if you experience graphics difficulties, especially when using OpenGL capabilities (shadows, bump maps, etc.). - Move the slider to the right to enable all accelerations.

Use this option	To do this
	Note: Slider settings are persisted to the registry so that you may continue with a “working” setting between sessions.
Recommended version	Use the optimal OpenGL level supported by your video card.

8. Click **OK**.

Using the 3D Performance toolbar

Overview of using the 3D Performance toolbar

There are three predefined performance settings on the **3D Performance** toolbar. You can change these settings or remove them, and you also can add two additional settings.

- Add a new performance setting button to the toolbar
- Rename a performance setting button
- Delete a performance setting button from the toolbar
- Move a performance setting button on the toolbar

Add a performance setting button to the toolbar

1. On the **3D Performance** toolbar, click **Edit Performance Settings** .
2. Click the **Toolbar** tab.
3. On the Toolbar page, click **New**.
4. In the **Add Performance Setting Button** dialog box, in the **Name** box, type a name for the new button.
5. Click **OK** twice.
6. To add more performance buttons, repeat the process.

Note:

You can specify up to five performance setting buttons.

Rename a performance setting button

1. On the **3D Performance** toolbar, click **Edit Performance Settings** .
2. Click the **Toolbar** tab.
3. On the Toolbar page, click **Rename**.
4. In the **Name** box, type the new name of the button.
5. Click **OK** twice.

Delete a performance setting button

1. On the **3D Performance** toolbar, click **Edit Performance Settings** .
2. Click the **Toolbar** tab.
3. On the Toolbar page, select a name from the **Performance Setting Buttons** list.
4. Click **Delete**.
5. Click **OK**.

Move a performance setting up or down on the list

1. On the **3D Performance** toolbar, click **Edit Performance Settings** .
2. Click the **Toolbar** tab.
3. On the Toolbar page, select a name in the **Performance Setting Buttons** list.
4. Click **Move Down** or **Move Up**

Adjusting interactivity

Interactivity is the speed and smoothness with which the model in the Viewing window reacts to your input.

When you work with complex models, the interactivity of your session may decline. When more parts are loaded and visible in the Viewing window, interactivity is slower.

Use these recommendations to increase interactivity:

- Set the **Fixed Frame Rate** option in the range of 5 to 10 frames per second. A higher number will result in a more responsive interactive performance, although you may see less of the model.

The frame rate is the rate at which your model is redrawn in the Viewing window. When you move a model that has many visible parts, you may notice decreased interactivity because of the time it takes your computer to redraw the model in its new position. By controlling the frame rate, the program draws only a portion of the model parts.

- Set these **Size Culling** options to control the minimum size of the parts that are drawn in the Viewing window:
 - Set screen size culling to cull parts with less than 1% estimated screen coverage.
 - Set model size culling to cull parts with a bounding box diagonal less than 1% the size of your entire model. (Enter this value in millimeters.)
- Adjust the **Image Quality**, or level of detail. Models are displayed in the Viewing window in varying levels of detail, depending on the distance of the model from the eye point. Parts that are farther from the eye point are displayed in a lower level of detail than are parts that are closer to the eye point.

Models that are displayed in a lower level of detail appear less complete, especially as the model is moving, but interactivity is improved.

- Control the manner by which parts are loaded and drawn in the Viewing window. Models with many parts can take several minutes to load.

You can:

- Wait until all the necessary parts are loaded before viewing the parts. With this option, the application is inactive, no new parts are visible in the Viewing window, and you cannot interact with the model until all the parts you have toggled on are loaded. At that time, all the parts appear in the Viewing window at the same time.
- Stop loading parts at any point by pressing the Esc key on your keyboard.
- Using certain advanced features may degrade your visual interactivity. Disable the following features if they are not necessary for your application:
 - **Shadows**
 - **Anti-aliasing**

- **Hidden line removal**
- **Two-sided lighting**
- Usually, the more memory your computer's graphics card has, the better the visual interactivity will be.

Troubleshooting graphics performance

If you experience graphics difficulties, especially when using advanced OpenGL capabilities (shadows, bump maps, etc.), you can use the OpenGL option in the 3D performance preferences to troubleshoot the problem.

Note:

Slider settings are persisted to the registry so that you may continue with a working setting between sessions.

1. In the Viewing window, right-click and choose **Performance→Edit**.
2. On the **Troubleshoot** tab, click and drag the **OpenGL** slider to change the effective OpenGL version used by the application.

Note:

- Dragging the slider to the left may make the application more stable but also compromise performance. Advanced visual effects also may be disabled as the OpenGL level is decreased.
- OpenGL 4.6 is the recommended minimum for Lifecycle Visualization graphics features.

3. Close and reopen the model for your changes to take effect.

Selecting parts

Part selection overview

When selecting parts and part features, you can:

- Select parts.
- Select all parts within a rectangular area or volume.
- Identify selected parts.

- Select part features like vertices, edges, and surfaces.

Select parts

Before you can perform tasks with individual parts of your 3D model, you must select the parts.

Do any of the following:

To	Do this
Select a part from the Viewing window	Click the part.
Select multiple parts	Press the Ctrl key as you click the parts in the Viewing window.
Select an assembly	In the Viewing window, click a part repeatedly. The first click selects the part; the second click selects the parent subassembly of the part. Each successive click selects one higher subassembly. Note: If you are viewing an assembly in Structure Manager or Manufacturing Process Planner, this behavior applies only if the entire assembly is loaded. If the entire assembly is not loaded, parent assemblies are selected on subsequent clicks only when the parent has loaded .jt parts. A parent subassembly is not selected if has no .jt parts loaded. Note: If you select a part encompassed by an end item, all other parts associated with the end item are also selected. To select individual parts within end items, right-click the end item in the assembly tree, and choose Lock/Unlock.
Select an entire model	In the Viewing window, right-click and choose Select All .
Deselect a part	Press the Ctrl key and click the selected part in the Viewing window.
Deselect all parts	In the Viewing window, right-click and choose Deselect All. -or- Click the background of the Viewing window.
Undo a selection action	Press Ctrl+Z.

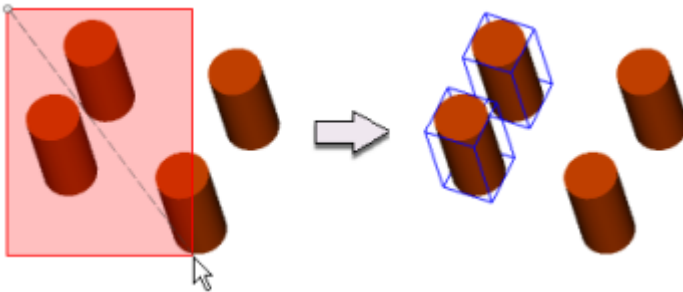
To	Do this
	Note: To reverse an undo, press Ctrl+Y.

Select parts in a rectangular area

You can define a rectangular area in the Viewing window and select parts within the rectangle.

1. On the **3D Selection** toolbar, click **Select Parts by Area** .
2. In the Viewing window, click and drag to the right or left to draw a rectangle around your parts. Control the part selection in the following ways:

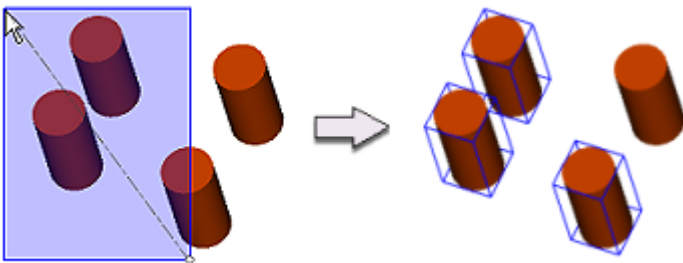
- Drag to the right to select parts that are completely inside the red rectangle.



Note:

For monolithic JTs, this action will select none of the components of a monolithic JT in the viewer.

- Drag to the left to select parts that are inside or intersect the blue rectangle.



Note:

For monolithic JTs, this action will select all the components of a monolithic JT in the viewer.

Note:

For monolithic JTs, **Select Parts by Area** function will select all or none of the components of a monolithic JT as per the direction of selection in the viewer unless you break it down by using **Show SubComponents** option in Structure Manager/Multi-Structure Manager/Manufacturing Process Planner.

To do that:

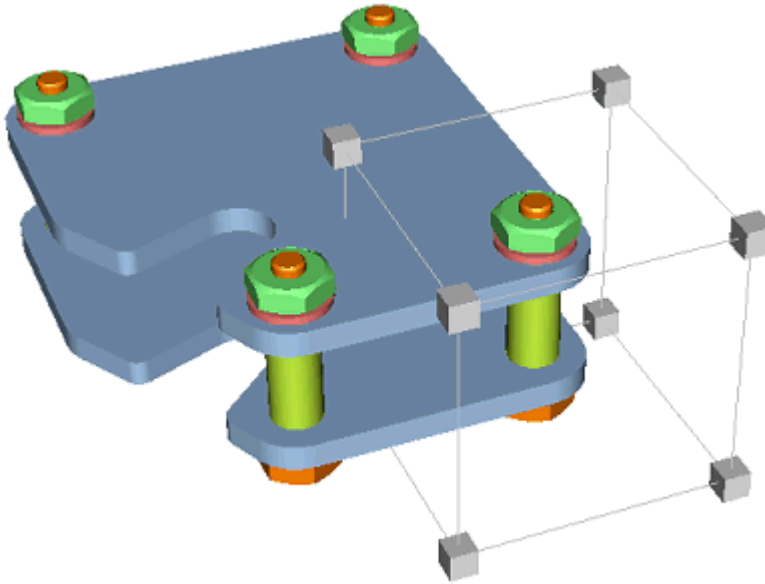
1. Select the BOMLine.
2. Click on the right mouse button.
3. Select **Show SubComponents**.
4. Use the **Select Parts by Area** function to make individual selection of the parts.

Select parts in a rectangular volume

You can define a rectangular volume in the Viewing window and select parts within the rectangle.

1. On the **3D Selection** toolbar, click **Select Parts by Volume** .
2. In the Viewing window, click a part and drag to the right or left to create a cube. Control the part selection in the following ways:
 - Drag to the right to create a cube that selects parts that are completely inside the cube. As you drag, the cube is red.
 - Drag to the left to create a cube that selects parts that are inside or intersect the cube. As you drag, the cube is blue.

The cube becomes a volume manipulator.



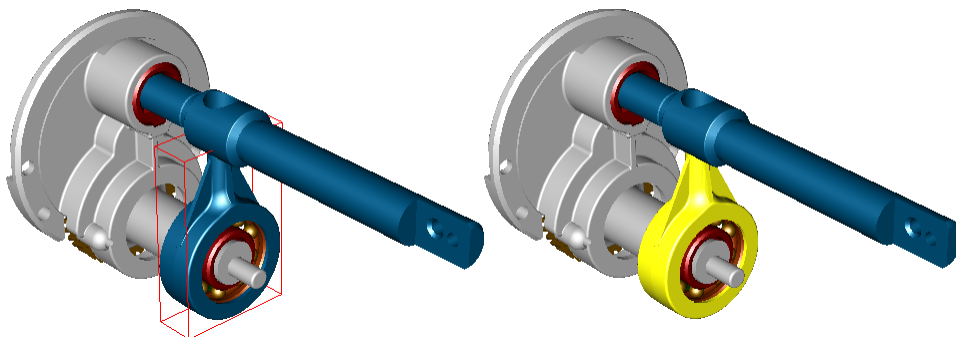
3. (Optional) Do the following to move and resize the volume manipulator:

To	Do this
Move the manipulator parallel to a face	Drag the face of the manipulator.
Move the manipulator perpendicular to a face	Hold Ctrl as you drag the face.
Constrain the move to one axis	Hold Shift as you drag a face.
Resize the manipulator uniformly	Drag a corner of the manipulator.
Resize the manipulator along one axis	Hold Ctrl as you drag a corner of the manipulator.

4. To complete the selection, on the **3D Selection** toolbar, click **Select Parts by Volume** .

Identify selected parts

Identify selected parts with a bounding box or with a color.



1. In the Viewing window, right-click and choose **Preferences**.
2. On the **Selection** tab of the **Preferences** dialog box, from the **Selection Style** list, select one of the following:

Select this style	To identify selected parts in this way
Bounding Box	Display a bounding box around selected parts.
Material Color	Highlight selected parts in a specified color.

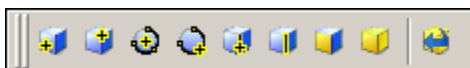
3. From the **Selection Color** list, select a color.
4. (Optional) Select the **Set as default** check box to use the selection style and color in all new 3D Viewing windows.
5. Click **OK**.

Selecting part features

Overview of selecting part features

By default, when you click a part in the 3D Viewing window, the entire part is selected. When you need to **select a part feature** to perform certain operations, such as measuring between two edges, the 3D Selection toolbar activates the commands that are available for the operation.

- See part features highlighted as you move the cursor over the model.
- Display the **QuickPick** dialog box to help you select the correct feature when there are multiple possible features.
- Filter your selection by choosing one or more part feature **types** on the **3D Selection** toolbar. For example, to measure between two edges, you can filter your selection so that only edges can be selected.



Use	To select
Point	Any point on a surface.
Vertex	An endpoint of an edge (or construction point geometry).
Arc Center	Pick the center of an arc.
Midpoint	Pick the midpoint of a line or arc.

Use	To select
Point on Edge	Pick a point on an edge.
Edge	A straight or curved edge (or construction line geometry).
Surface	A planar or non-planar surface.
Part	An entire part.

Tip:

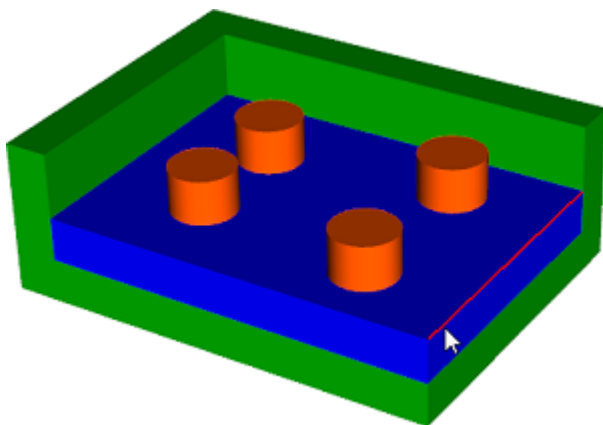
In Standard, Professional, and Mockup, when your 3D model is displayed with tessellation lines, you can see part features that would otherwise be hidden by the surface of the model. To pick through parts displayed with tessellation lines, you must activate **Pick Through Unshaded Parts**.

Select part features

When performing operations that require you to pick **part features**, like measuring between two edges, CAD-like tools help you pick the correct features.

1. Move the mouse pointer over the part.

Candidate part features are highlighted as you move over the part.

**Note:**

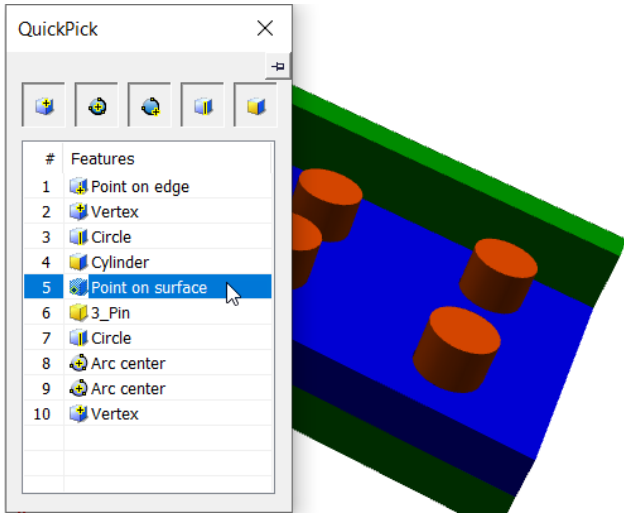
- If you do not see highlights, confirm that the **Display Highlights** preference is on.
- If you have older pre-JT v9.1 models with B-Rep data, you can **generate selection preview data** to improve selection preview highlighting.

2. (Optional) To specify the types of features to consider, do one of the following:

To	Do this
Limit the candidates to only one type of feature (for example, only edges)	On the 3D Selection toolbar, hold Ctrl and click the feature type.
Limit the candidates to only some types of features (for example, edges and surfaces)	On the 3D Selection toolbar, click the various feature types to turn them on or off. Note: You cannot turn off all types. If only one type is on and you attempt to turn it off, all types turn on.
To consider all types of part features	On the 3D Selection toolbar, click all the feature types to turn them on.

3. To select the part feature you want, do one of the following:

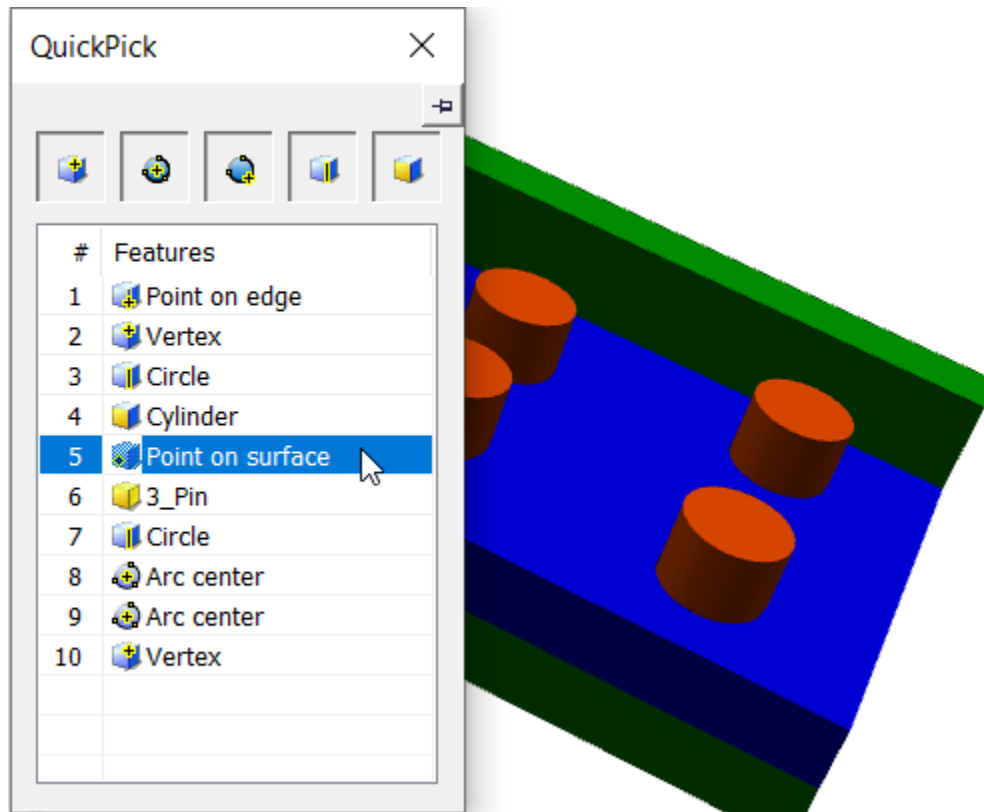
To	Do this
Pick the highlighted feature	Quickly pick the feature.
Pick from a list of part features near the mouse pointer	a. Stop moving the mouse pointer and wait for it to change to three dots . b. When the three dots appear next to the pointer, click to display the QuickPick dialog box. c. Choose the feature from the list.

To	Do this
	

QuickPick overview

The **QuickPick** dialog box helps you pick the correct part feature when there are multiple features near your mouse pointer. The **QuickPick** dialog box gives you a list of part features and highlights each feature in the Viewing window before you choose.

Move the mouse pointer near the part feature you want to select, pause for a moment, and then when three dots appear next to the pointer , click to display the **QuickPick** dialog box.



You can:

- Move down the list to see each part feature highlighted in the Viewing window, and then click to select the highlighted feature.
- At the top of the dialog box, click **Vertex** , **Arc Center** , **Midpoint** , **Edge** , and **Surface**  to control which types of features appear in the list.
- **Specify the time delay** before displaying the QuickPick prompt.
- Pin   the dialog box so it always opens in the same location.

Previewing part feature selection

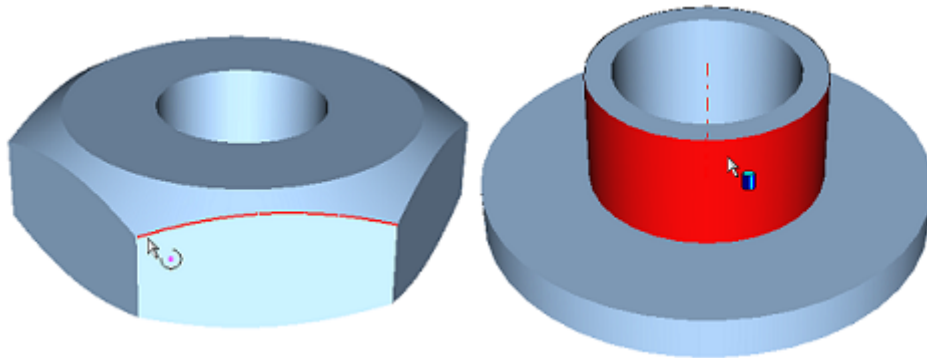
Overview of selection preview

For certain tasks, you must select part features rather than the entire part. Parts are composed of many features. Part feature *selection preview* helps you identify a part feature before you select it.

For example, if you need to select a specific edge, the closest edge appears highlighted as you pass the cursor over parts in the Viewing window. Click when the edge you want is highlighted.

There are three selection preview options. You can:

- Highlight the feature.
- Display the specific geometric types of edges and surfaces.
- Display construction lines for arcs, circles, and conic surfaces. Show center lines of circular edges and conic surfaces. For arc edges, show center lines and lines from the center to the end points of the arc.



Note:

- The Display Highlight option is on by default.
- If you have v9.1 JT or later data with B-Rep data, you can display selection preview Highlights, Geometric Types, and Construction Lines.
- A question mark symbol  next to the cursor when selecting part features indicates the parts do not contain the correct tessellation data for full selection preview.
 - If you have v9.0 JT or earlier data with B-Rep data, you can **generate selection preview data** from the B-Rep data on the fly.
 - If you have parts with no B-Rep data, you may see partial selection preview and the question mark symbol . You cannot generate the correct tessellation data for parts with no B-Rep data. Attempting to **generate selection preview data** displays a **Warning** dialog box with a list of parts with no B-Rep data.

Generate selection preview data

Note:

Your models must include B-Rep data for you to fully use part feature selection preview.

If your models include B-Rep data, you can use part feature selection preview with the following limitations:

- If you have v9.1 JT or later data, you can display selection preview Highlights, Geometric Types, and Construction Lines.
- If you have v9.0 JT or earlier data, you can display basic Highlights. You must generate selection preview data to display full Highlights, Geometric Types, and Construction Lines.

1. Start an operation that requires you to select a part feature (for example, 3D Measurement).
2. (Optional) To generate selection preview data for only some parts, do one of the following:
 - Select the parts that need selection preview data.
 - Turn off parts that do not need selection preview data.
3. On the **3D Selection** toolbar, click **Generate Selection Preview Data** .

The **Generating Selection Preview Data** dialog box displays the progress.

If you have parts that do not have B-Rep data, the **Warning** dialog box appears with a list of the parts.

Tip:

To reduce processing time and memory use when working with very complex models, generate selection preview data for selected parts, rather than entire models.

Set selection preview preferences

1. Do one of the following:
 - On the **3D Selection** toolbar, click **3D Selection Preferences** .
 - In the Viewing window, right-click and choose **Preferences**.
2. To use selection preview, in the **Preferences** dialog box on the **Selection** tab, select at least one check box.

Select this option	To
Display Highlights	Highlight the feature that is closest to the cursor.
Display Geometric Types	Identify the types of edges or surfaces.
Display Construction Lines	Display construction lines for arcs, circles, and conic surfaces.

3. Specify how to display highlighted points.
4. To change the color used for part feature highlighting and construction lines, select a color from the **Color** list.
5. (Optional) To turn off selection preview, in the **Preferences** dialog box on the **Selection** tab, clear all of the check boxes.
6. (Optional) Select the **Set as default** check box to save the preferences as the default behavior.

Selection previews with geometric type

You can display geometric types for edges and surfaces during part feature selection to help you select the correct feature. The type information appears on the status bar and as an icon next to the cursor.

Note:

- The parts must have B-Rep data.
- For v9.0 JT or earlier data, you must **generate selection preview data** for the parts.
- The selection preview preference **Display Geometric Types** must be selected.

This cursor	This on the status bar	Identifies
	Edge: Line	Linear edges
	Edge: Arc Edge: Circle	Circular arc edges
	Edge: Curve	General edges
	Face: Plane	Planes
	Face: Sphere	Spheres
	Face: Cone	Cones
	Face: Cylinder	Cylinders
	Face: Spline Surface	General surfaces

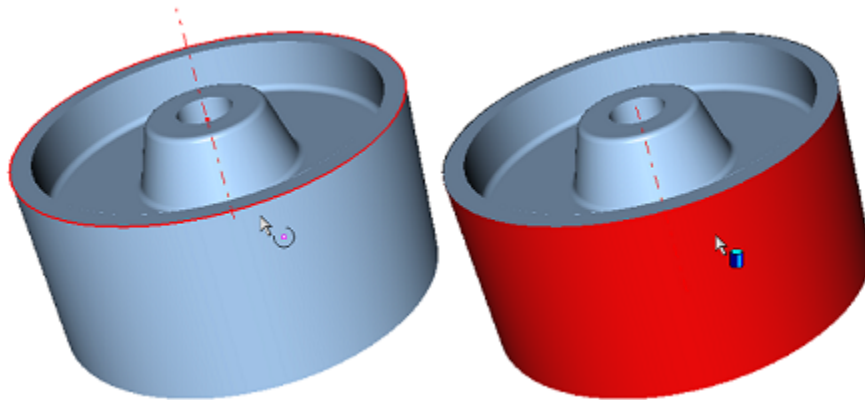
This cursor	This on the status bar	Identifies
	Face: Surface of Revolution	
	Preview data unavailable	Unknown part features

Selection previews with construction lines

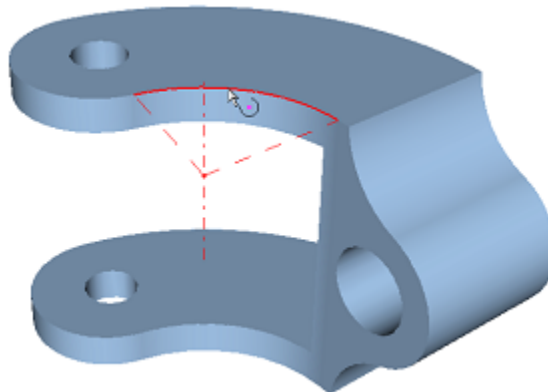
You can display construction lines for arcs, circles, and conic surfaces during part feature selection to help you select the correct feature.

Note:

- The parts must have B-Rep data.
- For v9.0 JT or earlier data, you must **generate selection preview data** for the parts.
- The selection preview preference **Display Construction Lines** must be selected.



Centerlines for circular edges and conic surfaces



Construction lines for arc edges

Preview part features to select

Use part feature selection preview to help you select the correct feature.

1. **Set selection preview preferences** to display at least one type of preview.
2. Start an operation that requires you to select a part feature.
3. **Generate selection preview data** if needed.
4. In the Viewing window, move the cursor over part features to preview the selection.
5. Click when the preview indicates that you are selecting the correct feature.

Calculating points

Overview of calculating points

There are several ways to specify points that are not part of the model geometry. You can:

- Enter coordinates specifying a **user defined point**.
- **Construct a midpoint** between two points.
- Calculate the **intersection** point of two linear edges.
- Find the **center of an arc** that passes through any three selected points.

Note:

- You can also set **measurement preferences** to display midpoint and extrapolated intersection points in the measurement display labels of some single and distance measurements.
- When using a calculated point for a single measurement, the measurement labels display information about the calculated point. You can use calculated points when measuring distance between points.

Enter coordinates of a point

1. On the **3D Selection** toolbar, click **User Defined Point** .
2. In the **Define Point** dialog box, type the **X**, **Y**, and **Z** coordinates and click **OK**.

Define Point ✕

Point Position:

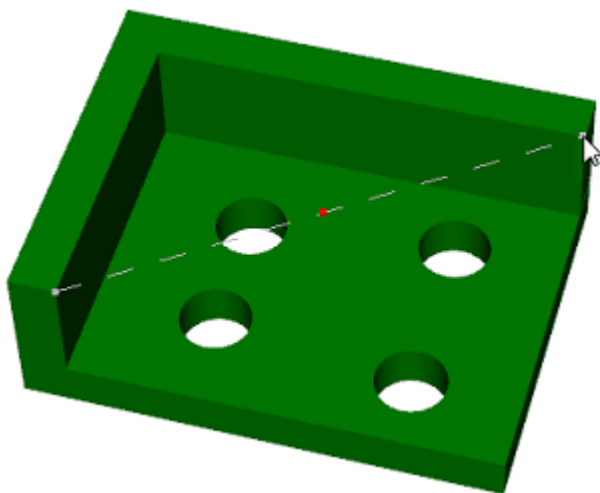
X:

Y:

Z:

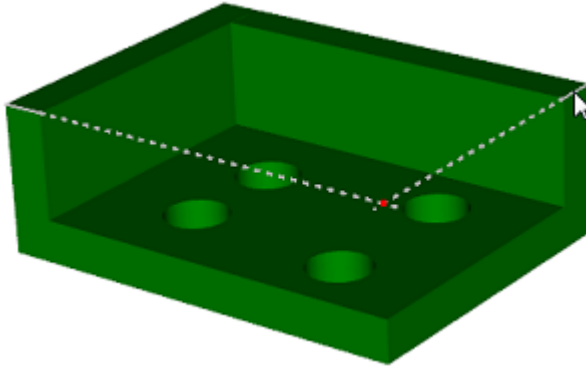
Construct a midpoint between two points

1. On the **3D Selection** toolbar, click **Construct Midpoint** .
2. Select **two points**.



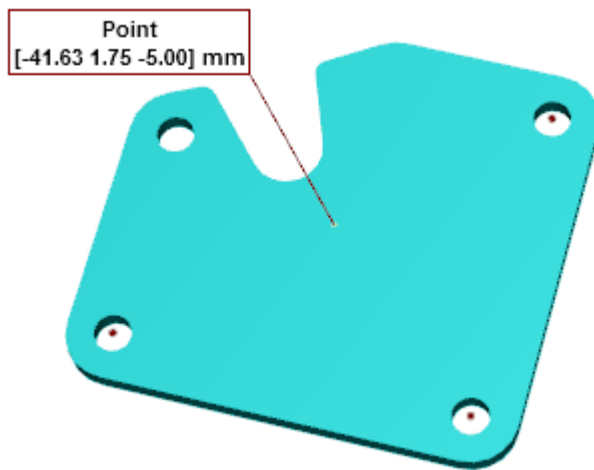
Calculate the intersection point of two edges

1. On the **3D Selection** toolbar, click **Intersection** .
2. Pick **two edges**.



Calculate the center of an arc formed by three points

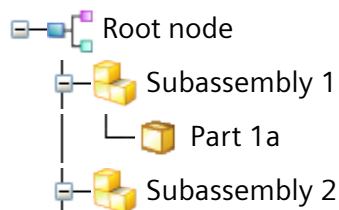
1. On the **3D Selection** toolbar, click **3Point Circular Arc** .
2. Pick **three points**.

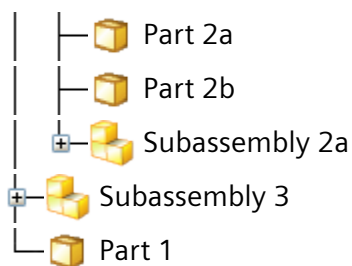


Managing assemblies

Overview of working with assemblies

When you open a model of an assembly, you can see a list of all its parts and subassemblies in the **Assembly** view. The list appears as an assembly tree. The assembly tree shows the assembly hierarchy, the logical grouping of the parts and subassemblies.





The highest level assembly is called the *root*. The root includes all the subassemblies and parts. Each subassembly can include other subassemblies, parts, or a combination of subassemblies and parts.

Each subassembly and part is called a *node*. The nodes directly below any subassembly are the *children* of that subassembly.

The lowest-level children are parts, which provide the geometry for the visual representation of your model in the Viewing window.

You can also create Alternate Hierarchies. The Alternate Hierarchy icons that appear on the Assembly page represent a copy of the original component.

The type of a node is indicated by its icon in the **Assembly** view. Next to that is a check box indicator of its **load and visibility status**.

This icon	Indicates
	The root node.
	An expandable node.
	A collapsible node.
	An assembly, subassembly, or end-item node.
	A subassembly that contains JtSimplification data.
	A subassembly that contains MMV data.
	A 4GD presented parent.
	A part or leaf node.
	A body or leaf node.
	A node designated as having no geometry.
	A monolithic JT node.
	An MMV node.

This icon	Indicates
	A locked assembly component.
	An end item.
	An overridden end item.
	An assembly component that cannot be clipped.
	A node in an alternate hierarchy that points to the original assembly structure of the model.
	A node in an alternate hierarchy that points to the original part structure of the model.
	An assembly node within an alternate hierarchy assembly.
	A part node contained within an alternate hierarchy assembly.

Load and visibility statuses of assembly nodes

When you view an assembly in the **Assembly** view, each **assembly node type** is indicated by its icon. Next to that is a check box indicator of its load and visibility status.

This check box	Indicates
	The part or assembly is not loaded.
	The part or assembly is loaded but not visible.
	The part or assembly is fully loaded and all geometry is visible. This symbol appears for an assembly even when some parts in the assembly intentionally have no associated geometry, as indicated by  (see below).
	The assembly is loaded and some of the parts with associated geometry are visible (assembly only). The assembly may have some parts for which the part geometry failed to load ().
	Some parts of the assembly are loaded and some of the parts with associated geometry are visible (assembly only).
	A file containing part geometry (.jt file) could not be opened. This symbol appears when the assembly references a .jt file that for some reason is not accessible; for example, it has been deleted.

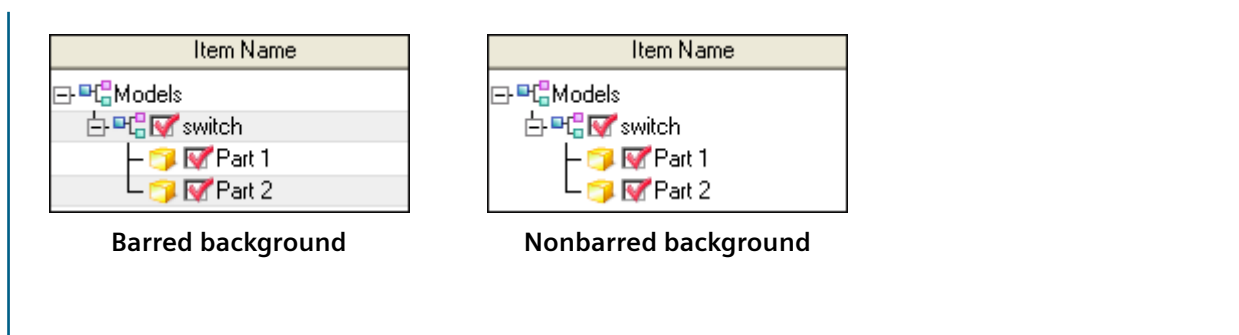
This check box	Indicates
	This symbol appears for parts with geometry that fails to load, not parts that are intentionally without geometry, as indicated by  (see below).
	No geometry (.jt file) is associated with the part. This represents parts for which no geometry has been added. This does not represent geometry that fails to load. For example, an assembly is opened from Teamcenter and the assembly references a new BOM line that does not have a JT representation. Either the file has not yet been translated or the item has only a placeholder for a part that has not yet been designed.

Specify assembly preferences

1. Access the **Assemblies Preferences** dialog box by doing one of the following:
 - Choose **Assembly→Preferences**.
 - In Base, in the assembly area, in an empty area to the left of assembly or part names, right-click and choose **Preferences**.
2. In Standard, Professional, and Mockup, to specify how a part is displayed in the Viewing window and highlighted in the assembly tree, select one or more of the following options:
 - **Models Tree**
 - **Active Alternate Hierarchy**
 - **Inactive Alternate Hierarchies**
3. Select or clear **Barred background** to specify the background appearance of the assembly tree.

Note:

The default setting is **Barred background**.



4. Select or clear **Show subset structure** to display 4GD worksets in the context of the presented parent hierarchy.
5. Select or clear **Show active partition scheme** to apply default partition schemes to all 4GD subsets.

This preference applies *active partition schemes* to subsets, for the opened document and any subsequently loaded documents. Active partition schemes are the default schemes set on subsets in the **Subset Defaults** attribute group under **Viewed Partition Scheme** in Teamcenter. For subsets that do not have an active partition scheme defined, no partitions are shown when the subsets are loaded.

Note:

If you select this preference, the partition scheme setting for all subsets is overwritten. Any partition schemes you have selected on individual subsets are cleared and the subset reverts to showing its active/default partition scheme, or no partition scheme if an active partition scheme is not defined for a subset.

Expand and collapse the assembly

To	Do this
Expand the entire assembly	On the Tree Control toolbar, click Expand All  .
	Note: If your model contains JtSimplification data, doing an Expand All loads all of the parts in the model.
Collapse the entire assembly	On the Tree Control toolbar, click Collapse  .
Expand the assembly one level	Click the + in the square to the left of the entry. -or-

	On the Tree Control toolbar, click Expand One Level  .
Collapse the assembly one level	Click the - in the square to the left of the entry. -or- On the Tree Control toolbar, click Collapse One Level  .

Tip:

Models translated from I-deas can include alternate part geometry that is used when displaying product and manufacturing information (PMI) with specific **model views**. When alternate part geometry is included in a model, it appears in the assembly listing separate from the original geometry. To find the original geometry for a part, right-click the alternate geometry in the assembly listing, and choose **Select Original**.

Lock or unlock an assembly

Some operations, such as selecting a part in the Viewing window, can expand the assembly automatically. You can lock assemblies to keep them from expanding.

In the **Assembly** view, right-click the assembly name and select **Lock/Unlock**.

A lock symbol  appears next to the name of a locked assembly node.

Search the assembly tree of a local .jt file

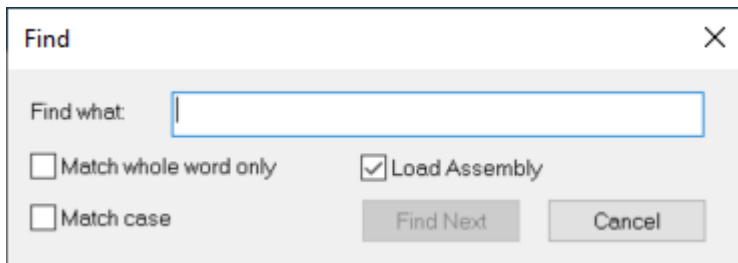
This topic describes the **Find** dialog, which allows you to search part and assembly names in the assembly tree of a .jt file that was opened from the local file system. Different versions of this dialog are available, depending on the service level. If you are using Mockup, you can also use the **3D advanced search functionality**, which allows you to find parts and assemblies using names or metadata and display the matching results in the Viewing window using a visual report.

Base, Standard and Professional

The number of search options in the **Find** dialog box varies for the different viewer service levels. The following describes how to search the assembly tree if you are using Base, Standard, or Professional.

1. Open the **Find** dialog box in one of these ways:
 - Right-click in the assembly tree and choose **Find**.
 - On the **Tree Control** toolbar, click **Find** .

The **Find** dialog box appears.



2. (Optional) Check the **Match whole word only**, **Match case**, or **Load Assembly** check boxes.
3. In the **Find what** section, enter a text string to find.
4. Click **Find Next**.

The first entry that contains the search text string is selected. It is highlighted in the assembly tree. Click **Find Next** to find other entries.

Mockup

There are more search options at the Mockup viewer service level. The following describes how to search the assembly tree if you are using Mockup.

1. Open the **Find** dialog box in one of these ways:
 - Right-click in the assembly tree and choose **Find**.
 - On the **Tree Control** toolbar, click **Find** .

The **Find** dialog box appears.

When you search the assembly structure of a .jt file that was opened from the local file system, the following options are selected as the default settings: **Text in Assembly Workspace**, **All**, and **Load metadata as needed**.

2. In the **Target** section, select **All** to search all parts and assemblies, or select **Visited** to search all visible parts.
3. (Optional) Check the **Match whole word only**, **Match case**, or **Load metadata as needed** check boxes.
4. In the **Find what** section, enter a text string to find.
5. Click **Find Next**.

The first entry that contains the search text string is selected. It is highlighted in the assembly tree. Click **Find Next** to find other entries.

Searching assemblies saved in Teamcenter

Overview of searching assemblies

The search process for objects in an assembly depends on where the assembly is saved. When you work in the standalone application viewer, assemblies are saved to the local file system. When you work in Teamcenter, assemblies are saved in the server.

When you search for parts or assemblies that are saved to the local file system, all the parts and assemblies must first be loaded before the search begins. This search is also known as a cache search.

The 'Find' dialog box is shown with the following settings:

- Find What:** An empty text input field.
- Target:**
 - ☒ Text in Assembly Workspace
 - ☐ Teamcenter
- Scope:**
 - ☒ All
 - ☐ Visited
- Options:**
 - ☐ Match whole word only
 - ☒ Load metadata as needed
 - ☐ Match case
- Information:**

Text based search in Assembly Workspace for All items.
- Buttons:** 'Find Next' and 'Cancel'.

Note:

For local file system searches, notice that **Text in Assembly Workspace** is selected.

When you search for parts and assemblies that are saved to the Teamcenter database, only matching results are loaded in the viewer. This search is known as a cacheless search.

The 'Find' dialog box is shown with the following settings:

- Find What:** An empty text input field.
- Target:**
 - ☐ Text in Assembly Workspace
 - ☒ Teamcenter
- Type:**
 - ☒ Item ID
 - ☒ Item Name
- Options:**
 - ☐ Match whole word only
 - ☒ Load metadata as needed
 - ☐ Match case
- Information:**

Teamcenter Search for Item ID or Item Name.
- Buttons:** 'Find Next' and 'Cancel'.

Note:

- For cacheless searches, notice that **Teamcenter** is selected. You can then choose to search for **Item ID**, **Item Name**, or both of these objects.
- Because the assembly is saved in Teamcenter, one of the important benefits of using cacheless searches, unlike a cache search, is the entire assembly is not loaded. The cacheless search is more efficient because the search is processed on the server-side and only the requested search results are downloaded to the local cache.

The following filter settings also support cacheless searches:

- **Attribute filters**
- **Clearance filters**
- **Name filter**
- **Volume filters**

You can also use **Visual reports** to search for parts and assemblies stored in Teamcenter. That is, visual reports support cacheless searches.

Note:

- Generating a cacheless search for data stored in Teamcenter is only supported at the Mockup service level.
- Generating a cacheless search for data stored in Teamcenter using filters is only supported at the Mockup service level.

Search an assembly saved in Teamcenter

Use **Find** to locate a part or assembly in a model that is saved in Teamcenter and opened in the Lifecycle Viewer or in the integrated standalone application viewer. You can search for **Item ID** only, **Item Name** only, or both of these objects.

1. From My Teamcenter, send an assembly to the Lifecycle Viewer or to the integrated standalone application viewer.
2. Open **Find** in one of these ways:
 - Right-click on a specific component and select **Find**.
 - Right-click anywhere in the Assembly tree and select **Find**.

- Choose **Tools**→**Assembly**, and when the **Assembly** menu appears, choose **Assembly**→**Find**.

The **Teamcenter** option is only available when the **Find** dialog is accessed by right-clicking on a component that was loaded from Teamcenter.

- (Optional) In the **Type** section, check the **Item ID** and the **Item Name** check box, or only check one of them.

When both check boxes are checked, the processing searches for an item ID and an item name.

If you know the general nature of the search, checking only the **Item ID** or **Item Name** check boxes may improve the search performance.

Example:

Search string conditions are very flexible. When you select **Item ID** and **Item Name** and then load subassembly 33000048/A:1-drag-assembly the following search inputs are valid:

- 33000048
- drag-assembly
- drag

- assembly
- assem
- 33000
- 0048
- 3000

The text string before /A is the **Item ID** and the text string after /A is the **Item Name**. The symbol /A indicates that this is Revision A of this object.

The wild card character (*) is automatically added to the beginning and the end of the string. If you do not want to add the wild card, check the **Match whole word only** check box in the **Find** dialog box.

4. Click **Find Next**.

The first entry that contains the text string is selected. Click **Find Next** to find other entries.

Using cacheless search with filters and visual reports

Cacheless search using filters

The following filter settings support cacheless searches:

- **Attribute filters**
- **Clearance filters**
- **Name filter**
- **Volume filters**

Note:

Cacheless searches are not supported with Workset or 4GD workset data.

Cacheless search with visual reports

For a **visual report** saved in Teamcenter, you can **generate a cacheless search** for assembly and part attributes.

An example of applying and undoing filter search criteria

All filters support applying filter changes and then undoing those changes back to the original position. The benefit of applying changes and then undoing them is to visually display the effect of using filter conditions.

Example:

The *garage_door_opener.jt* assembly is used in this example. Also, the **Name filter** is used to display just the **Motor** part of the assembly. Next, apply and revert options are selected to display just the *Motor.asm_31* part and then revert to displaying the entire fishing reel assembly.

1. Open *garage_door_opener.jt* and make sure the entire assembly is visible.
2. Choose **Menu** → **Actions** → **Filters**.
3. Select **Name Filter**, and then select **Show Only** from **Filter Output Action**.
4. Click **Properties**, enter **Motor** in **Specify name, use (*) as wildcard (any length), or (?) (zero or one characters)**, and then click **OK**.

You can use *** or *?* as **wildcards** in any position.

5. In **Filter Manager**, click **Apply**.
Only the spool part is displayed.
6. In **Filter Manager**, click **Revert**.
The entire fishing reel assembly is displayed.

This example demonstrates how applying and reverting filter results affects all filters.

Working with assembly columns

Overview of working with assembly columns

Add columns to the **Assembly** view based on metadata associated with the part. Assembly columns enable you to filter information based on part attributes.

When you work with columns, you can

- Add new assembly columns.
- Reorder assembly columns.
- Hide assembly columns.

- Adjust the width of assembly columns.

Add new assembly columns

1. In the **Assembly** view, right-click a part, and select **Properties**.
2. In the **Properties** dialog box, select an attribute from the list.

Note:
You can add new columns only if the part has attributes.

3. Click **Add as Column**.
4. Click **OK**.

The attribute you selected appears as a column in the **Assembly** view.

Reorder assembly columns

1. Right-click a column heading and choose **More**.
2. In the **Manage Columns** dialog box, select a column, and click **Move Up** or **Move Down**.
3. Click **OK**.

Hide assembly columns

1. Right-click a column heading.
2. In the column list, clear the check box for the column that you want to hide.

Adjust the width of assembly columns

1. Move the cursor over the right edge of the column that you want to adjust.
2. When the cursor changes to an icon of two arrows separated by a single line (), click and drag the column edge left or right to decrease or increase the column width.

Note:
A hidden column has a width of zero. If you have hidden columns in your assembly, when you move the cursor over a column edge it may change to an icon of two arrows separated

by two lines (). Click and drag the column edge using this cursor to adjust the width of the hidden column.

Working with leaf components

Overview of working with leaf components

Leaf components are equivalent to the CAD leaf component, and CAD concepts like layers are supported within leaf components. You can view a leaf component as a single node in the **Assembly** view or view it expanded, with layer definitions.

You can automatically set leaf components to appear in the **Assembly** view when you open a 3D model, or manually display the leaf structure after you open the model.

Note:

You can use the **Layer Filter** dialog box to configure which **layers** are enabled in a leaf component.

Display leaf components automatically

1. Choose **File**→**Preferences**→**3D Loader**.
2. On the Other page, select **Show leaf component structure**.

When you load a file, leaf components will be displayed in the **Assembly** view.

Show or hide leaf components

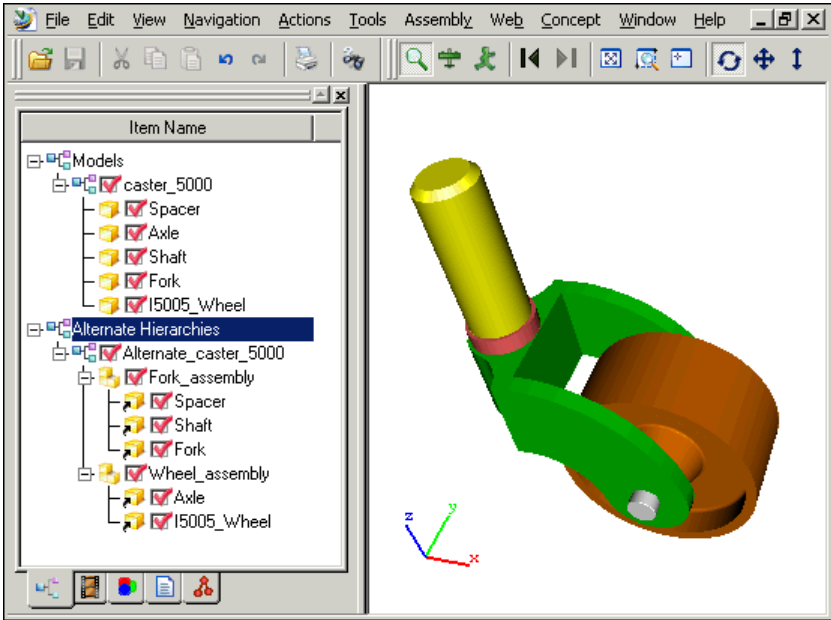
1. Open a 3D model that contains leaf structure.
2. In the empty white area to the left in the **Assembly** view, right-click and select **Show Leaf Structure**.

Leaf components are displayed in the **Assembly** view.

Working with alternate hierarchies

Overview of working with alternate hierarchies

You can create one or more alternate hierarchies that show an assembly with the parts grouped and organized to suit your needs (for example, by size, color, or supplier). Alternate hierarchies appear in the **Assembly** view.



Create alternate hierarchies

- 1. In the **Assembly** view, right-click the empty area to the left of the assembly, select **Create Alt Hierarchy**, and select or clear **Preserve Structure**.
 - If you select **Preserve Structure**, the alternate hierarchy is based on the model assembly hierarchy.
 - If you clear **Preserve Structure**, and the alternate hierarchy is based on the hierarchy of parts in the model.

Note:

Preserve Structure only applies to creating alternate hierarchies when you select one of these options:

- **From Visible**
- **From Selected**

- 2. In the **Assembly** view, right-click the empty area to the left of the assembly, select **Create Alt Hierarchy**, and then choose one of the following options:

Use this option	To
New	Create an empty alternate hierarchy.
From All	Create an alternate hierarchy that is a duplicate of the original assembly.

Use this option	To
From Visible	Create an alternate hierarchy of the visible parts.
From Selected	Create an alternate hierarchy of the selected parts.

A new top-level entry, with the default name **Alt. Hierarchy**, appears in the **Assembly** view.

- (Optional) To rename the alternate hierarchy, right-click **Alt. Hierarchy** and select **Rename**.
- Enter the new name.

Specify how alternate hierarchy parts are highlighted

Once you create one or more alternate hierarchies, you can specify how you want to display parts on the Assembly page. There are three options to choose from.

- Choose **Tools**→**Assembly**→**Preferences**.
- In the **Assembly Preferences** dialog box, select any of the following options:

Select	Then do this
Model Trees	Click one or more parts in the Viewing window. On the Assembly page, your parts are highlighted in the original model.
Active Alternate Hierarchy	- Click the top most node of the alternate hierarchy that you want active. - Right-click and select Active. - Click one or more parts in the Viewing window. On the Assembly page, your parts are highlighted in the active alternate hierarchy.
Inactive Alternate Hierarchies	(If you have an active alternate hierarchy) Click the top node of the active alternate hierarchy, then right-click and select Active. The active alternate hierarchy is now inactive. - Click one or more parts in the Viewing window. On the Assembly page, your parts are highlighted in inactive alternate hierarchies.

Modifying parts in alternate hierarchies

Add parts to an alternate hierarchy

1. In the **Assembly** view, select one or more parts, right-click, and choose **Copy**.
2. Right-click the target alternate hierarchy and choose **Paste**.
3. (Optional) In the **Assembly** view, select one or more parts in the original assembly list or in an alternate hierarchy and drag the selected items to the designated hierarchy.

Note:

- You can copy items from either the original assembly or any alternate hierarchy, but you can only cut items from alternate hierarchies.
- Hold the Ctrl key to select multiple items.
- You cannot copy parts between alternate hierarchies by dragging them.

Create a placeholder in an alternate hierarchy

Right-click the alternate hierarchy and choose **Add New**.

A new placeholder, with the default name *New Item* is added to the alternate hierarchy.

Connect a part in an alternate hierarchy to a part in the original assembly list

1. Select a part in the Viewing window.
2. Right-click a part in an alternate hierarchy and choose **Connect**.

The two parts are connected.

Rename parts in an alternate hierarchy

1. In the **Assembly** view, right-click a part in an alternate hierarchy and choose **Rename**.
2. Enter the new name.

Delete parts in an alternate hierarchy

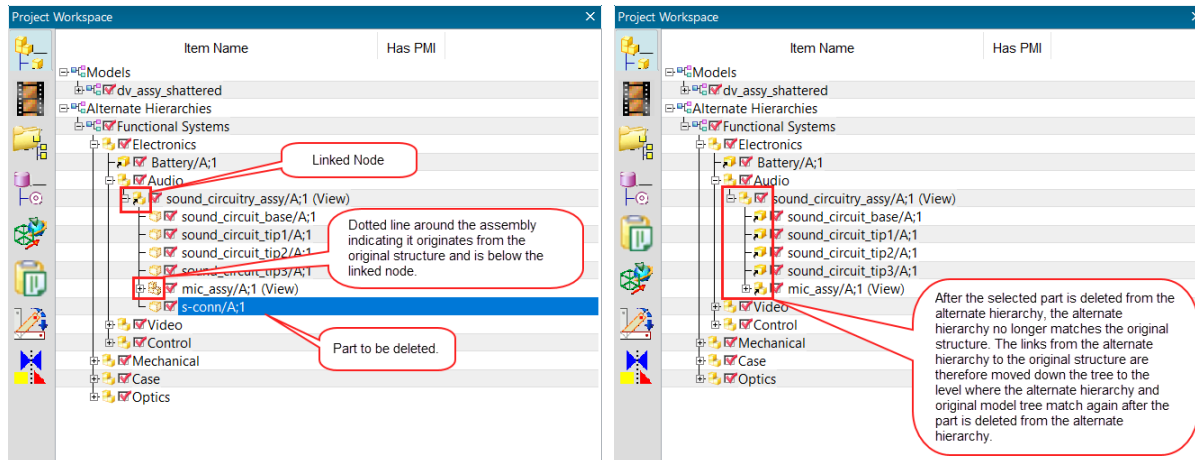
- Right-click a part in an alternate hierarchy and choose **Delete**.

You can use the **Undo** command to undo the last **Delete** action.

Deleting a linked node in an alternate hierarchy

When there is a linked node from the alternate hierarchy to the model, the structure below the linked node originates from the original model. Deleting a structure that is under a linked node removes it from the alternate hierarchy but does not remove it from the original structure.

See [Overview of working with assemblies](#) for information about assemblies and assembly icons.



Viewing model layers

Overview of viewing model layers

You can load 3D models containing layer information and filter part visibility based upon the layer settings within the file. Layer contents can include parts and PMI markups.

If a model file contains a **Default Layer Filter** (the layer filter that was active within the CAD system before translation), you can specify that it be applied when you load the model. You can also apply the **Default Layer Filter** after you load the model through the **Layer Filter** dialog box.

If PMI data is present in the file, you can also choose to display markups that are associated with layer settings.

Apply the Default Layer Filter automatically

1. Choose **File**→**Preferences**→**3D Loader**.
2. On the Other page, select **Set Default Layer Filter**.
3. Load your model.

Tip:

In the **Layer Filter** dialog box, select the default filter to apply after the model is loaded. For added control, select **Use default filter when selected filter is not available**. This option forces leaf components that do not have the current filter available to use the default filter rather than showing all component bodies.

Filter part visibility using layer filters

1. Choose **Actions**→**Layer Filter**.
2. In the **Layer Filter** dialog box, in the **Available Filters/Layers** section, select a filter.

The layers contained in the filter are also listed.

Note:

If this is the first time you are using the **Layer Filter** dialog box for this file, you will need to **Find Layers**.

3. (Optional, if PMI data is associated with layer information present in the model) Select the **Show PMI** check box.
4. If you want to display all of the components in the filter, select the **Show all components in applied filters** check box.
5. If you want to select all of the components in the filter, select the **Select all components in applied filters** check box.
6. If you want to apply the default filter when the selected filter is unavailable, select the **Use default filter when selected filter is not available** check box.
7. Click **Apply**.

The layer filter is applied to the model in the Viewing window.

8. Click **OK**.

Gather layer filter information

You can gather layer information from all parts, parts you selected, or just from the visible parts of your model.

1. Load the model.

2. Choose **Actions**→**Layer Filter**.
3. In **Search Scope**, select one of the following:
 - **All**
 - **Selected**
 - **Visible**
4. Click **Find Layers**.

Manually apply and show selected layers

1. Choose **Actions**→**Layer Filter**.
2. Clear **Show all components in selected filters**.
3. In **Available Filters/Layers**, select a layer.

Tip:

Press Ctrl to select multiple layers one at a time; press Shift to select a range of layers.

4. Click **Apply**.

Information present on the selected layer(s) appears in the Viewing window.

Note:

When you apply a layer filter, all parts that do not have layers associated with them retain their current visibility state.

Working with end items

Viewing end items

You can view a structure containing end items in Teamcenter lifecycle visualization. An end item is the top-level node of an assembly or subassembly that typically represents a complete product or process. Structure nodes are specified as end items in Manufacturing Process Planner.

When a structure containing an end item is opened in Lifecycle Visualization, the end item is displayed as a single line in the assembly tree. When an end item is selected in the tree, or when a part within an end item is selected in the 3D Viewing window, all of the other parts encompassed by the end item are also selected. You can override an end item to display its hidden structure or to select individual parts.

End items are identified in the assembly tree with the following icons:

 — An end item

 — An overridden end item

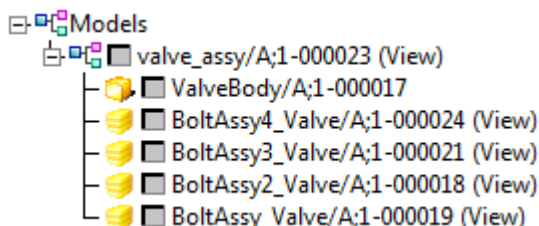
Note:

- The end item expansion state does not persist in PLM XML saved from Lifecycle Visualization.
- Product views may capture the state of expanded end items, depending on the model state. For example, if a product view is captured while an individual part is selected in an expanded end item, applying the product view expands the end item to restore the part selection.
- The following operations can override end items:
 - Running visual reports.
 - Searching the assembly.
 - Performing clearance analysis when the clearance element type is set to **Part** or **Leaf Component**.
- End items cannot include JtSimplification or Massive Model Visualization (MMV) data.

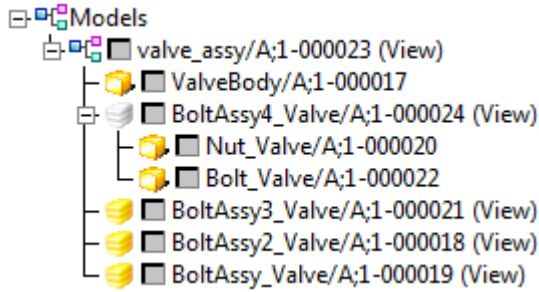
Display suppressed structure in end items

In the Lifecycle Viewer or the standalone application viewer, you can override end items to display suppressed structure.

1. Send visualization data containing end items to the Lifecycle Viewer or the standalone application viewer.
2. In the assembly tree, right-click the top-level node, and select **Expand**.



3. Right-click an end item, and choose **Lock/Unlock**.
4. Expand the structure encompassed by the end item.



Override end items automatically

You can specify for the application to override end items when 3D models are opened.

Note:

The **Show end item structure** setting is applicable only for structure saved locally as PLM XML and loaded in the standalone application viewer.

1. Choose **File**→**Preferences**→**3D Loader**.
2. Click the **Other** tab.
3. In the **General** section, select the **Show end item structure** check box.
4. Click **OK**.
5. Open a locally-saved PLM XML file containing structure with end items.

End items in the assembly are overridden. You can view and manipulate individual structure nodes within the assembly end items.

Choosing reference sets

Reference sets overview

NX reference sets are named configurations of geometry applied in the context of an assembly. A single component JT file contains all of the information necessary to define the geometry configurations for each reference set.

The loaded reference set determines what geometry is active for a given component. This geometry is used for the purposes of visualization of the component, as well as all analysis operations. Component bodies that do not belong to the loaded reference set are unavailable in the assembly tree.

You can specify the order in which the application attempts to load reference sets when a model is opened. You can also dynamically change the active reference set at any time during a work session.

Note:

Support for reference sets has the following limitations:

- The application of reference sets on an assembly level may differ between Teamcenter lifecycle visualization and NX. In Teamcenter lifecycle visualization, all reference sets are applied on leaf components, so applying a reference set on an assembly amounts to applying it on all leaf components in that assembly. In NX, additional reference set logic may apply on the assembly level itself. This difference in behavior is as designed.
- Support of the **Empty** reference set within Lifecycle Visualization is limited and may not exactly reproduce its treatment in NX.
- Reference set transforms are not supported.
- Component exclusions with reference sets are not supported.
- PMI references with reference sets are not supported.
- If a reference set contradicts an applied layer filter for any given part, the reference set is not applied to the part. Likewise, if a layer filter contradicts an applied reference set for a part, the layer filter is not applied to the part.
- Reference sets are not supported in NX part files opened in Teamcenter lifecycle visualization. To view the reference set, convert the model to a JT file.

Change the load order for reference sets

You can specify the order in which the application attempts to load reference sets when a model is opened. The first reference set in the list that is found in the model is loaded.

Note:

The reference sets options in the **3D Loader Preferences** are applicable only to the standalone application viewer and the Lifecycle Viewer. To specify the reference set load order for embedded viewers within Teamcenter applications such as Structure Manager, you must use the **JT_FILE_PRIORITY_REFSETS** Teamcenter preference.

1. Choose **File**→**Preferences**→**3D Loader**.
2. In the **3D Loader Preferences** dialog box, click the **Reference Sets** tab.
3. Do any of the following:

To	Do this
Change the reference set load order.	- In the Reference Set Load Order list, select the name of a reference set. - Click Move Up or Move Down.
Add a reference set to the load order.	- Click Add. - In the Add dialog box, type the name of the reference set. - Click OK.
Remove a reference set from the load order.	- Select the name of a reference set. - Click Remove. Note: The following default reference sets cannot be removed from the load order: • [Model] • [Entire Part] • [Empty] • [As Saved]
Load the default reference set according to the order specified in the Teamcenter JT_FILE_PRIORITY_REFSETS preference.	Select the Teamcenter Reference Set settings take precedence check box.

4. Click **OK**.

The next model you open that contains reference sets will load according to these settings.

Change the active reference set

You can dynamically change the active reference set from the assembly tree or the Viewing window.

- In the **Assembly** view or in the Viewing window, right-click a part or assembly, point to **Replace Reference Set**, and select the reference set you want to load.

Note:

For performance reasons, the list of available reference sets may not include every reference set present in your data. Click **Selection** to display the **Reference Sets** dialog box, which you can use to search your parts for additional reference sets.

Search parts for reference sets

Use the **Reference Sets** dialog box to identify all of the reference sets present in your data.

1. Choose **Actions→Reference Sets**.
2. In the **Reference Sets** dialog box, from the **Search Scope** list, select one of the following options to specify the scope of the search:
 - **All**
Search all parts.
 - **Selected**
Search all selected parts.
 - **Visible**
Search all visible parts.
3. Click **Find Reference Sets**.
4. If you want to show or hide the names of the default reference sets, select or clear the **Show predefined reference sets** check box.
5. From the **Apply to** list, select one of the following options to specify the scope of the reference set application:
 - **All**
Applies the reference set to all parts.
 - **Selected**
Applies the reference set to all selected parts.
 - **Visible**

Applies the reference set to all visible parts.

6. Click **Apply**.

Note:

When a reference set is applied, all parts that cannot change to the reference set are listed in the **Change Reference Set** dialog box. If you want to investigate such parts, click **Select Parts** to select the parts, or else click **Close** to close the dialog box.

Control the active reference set in layer filters

You can apply reference sets to the contents of a layer filter.

1. Choose **Actions**→**Layer Filter**.
2. In the **Layer Filter** dialog box, in the **Available Filters/Layers** section, select a filter.
3. Select the **Select all components in applied filters** check box.
4. Click **Apply**.

The layer filter is applied to the model, and the parts are selected.

5. The reference set is applied to the selected parts.

In the **Assembly** view or the Viewing window, right-click a selected part, point to **Replace Reference Set**, and select the reference set you want to apply.

Note:

If a reference set contradicts the applied layer filter for any given part, the reference set is not applied to the part. Likewise, if a layer filter contradicts an applied reference set for a part, the layer filter is not applied to the part.

Add the reference set column to the assembly tree

You can specify to display the name of the active reference set in the assembly tree.

- In the assembly tree, right-click a column header, and choose **Reference Set**.

Item Name	Reference Set
Models	
model	
main_assy	
view_assy/A;1 (View)	
side_cover/A;1	MODEL
side_plate/A;1	MODEL
back_plate1/A;1	MODEL
rear_flux_assy/A;1 (View)	
circuit_assy/A;1 (View)	
capacitor_assy/A;1 (View)	
top_level_assy/A;1 (View)	
case/A;1 (View)	
spool_assy/A;1 (View)	
connector/A;1	MODEL

Save the reference set state in snapshots

You can use snapshots to preserve the reference set state.

1. On the **Snapshots** toolbar, click **Preferences** .
2. In the **Snapshot Capture Preferences** dialog box, ensure the **Reference Sets** check box is selected.
3. Click **OK** to close the **Snapshot Capture Preferences** dialog box.
4. On the **Snapshots** toolbar, click **Add** .
5. If you want to save and restore your work in a subsequent work session, save a session or PLM XML file.

Simplifying complex JT data

Overview of JtSimplification data

Complex JT models accessed remotely from Teamcenter often suffer from long loading times because of the size and complexity of their data. JtSimplification datasets solve this problem by providing simplified data for each of a model's subassemblies. When you open a JtSimplification dataset and turn on the root node of the model, the simplified data is automatically loaded and displayed, which enables you to view the entire 3D model much more quickly than is otherwise possible with a standard JT file. If you choose to explore the model in greater detail, the full data for each subassembly level is loaded incrementally as the structure is expanded in the assembly tree or in the viewing window.

JtSimplification datasets are created using the **Simpngen** translator, which is included with the Teamcenter Dispatcher system. Administrators can schedule JtSimplification production from the My Teamcenter **Translation** menu.

JtSimplification data is used automatically when you send the assembly into the viewer and turn on the root node. Since the geometry from a standard JT subassembly and a simplified subassembly looks the same in the viewing window, a gray icon  is used to represent subassemblies that consist of simplified geometry.



Note:

JtSimplification data requires an additional license and is supported only in the standalone viewers and the Lifecycle Viewer within the Teamcenter Rich Client. If you do not have a license for the JtSimplification functionality or if you load a JtSimplification dataset in an unsupported viewer, the full model loads as standard JT data.

Using JtSimplification data with other features

When you load a JtSimplification dataset, the Viewing window displays simplified data for each of the model's subassemblies, which may lead to unexpected results when you perform certain actions within the viewer. The table below lists features that may exhibit non-standard behavior when used with JtSimplification data.

Tip:

To avoid any of the limitations described below, **resolve the simplified data** to fully load the assembly information and geometry.

Feature	Behavior with JtSimplification data
3D file export	If you export a JtSimplification dataset, subassembly nodes that contain JtSimplification data are not included in the exported model.
Snapshots	When you capture a snapshot with a JtSimplification dataset, the simplified data is not resolved and the snapshot image is produced from the simplified geometry. Tip: You can apply a snapshot that captured simplified geometry even after the geometry is resolved.
Session save	When you save a session file with the Capture static structure check box selected in the Session Save As dialog box, all of the JtSimplification geometry is resolved as part of the saved session. Otherwise, the state of unresolved simplified geometry is maintained.
Part or assembly properties	When you right-click a part or assembly and select Properties , the JtSimplification data is not resolved and the displayed geometric properties are computed from the simplified geometry.
3D measurement	When you perform 3D measurements on JtSimplification datasets, the simplified geometry is not automatically resolved.
Clearance	When you perform clearance analysis with JtSimplification datasets, the simplified geometry is not automatically resolved.
Part Edit	If you perform any Part Edit operation, all JtSimplification geometry is automatically resolved and all of the parts are loaded.
Cross section	If you perform any cross section operation, all JtSimplification geometry is automatically resolved and all of the parts are loaded.
Show edges	If you right-click a subassembly and select Show Edges , the JtSimplification geometry of the selected subassembly is automatically resolved.
Outline capture	If you perform an outline capture on a JtSimplification dataset, the simplified geometry is used to create the outline capture.
Concept appearance edit	If you perform any Concept appearance edit operation, all JtSimplification geometry is resolved and all the parts are loaded.
True Shading	If you apply occurrence specific materials to simplified geometry, the affected subassembly is resolved and fully loaded. Also, global wash and metallic material finishes may not display as expected.
PLM XML save	When you save a JtSimplification dataset as PLM XML, all JtSimplification geometry is automatically resolved and all the parts are loaded.

Resolve and fully load JtSimplification data

When you open a JtSimplification dataset and turn on the root node of the model, the simplified data is automatically loaded and displayed. If you want to view the contents of a subassembly in more detail, you can manually expand the simplified structure to fully load the assembly information and geometry.

Do any of the following:

To	Do this
Resolve all of the simplified data within the model	In the assembly tree, right-click the root node of the model and select Resolve All .
Resolve the next level of simplified data within a single subassembly	In the assembly tree, click the plus sign  to the left of the subassembly node . -or- In the assembly tree, right-click the subassembly node and select Resolve. -or- In the Viewing window, right-click geometry within the subassembly and select Resolve.
Resolve all levels of simplified data within a single subassembly	In the assembly tree, right-click the subassembly node and select Resolve All. -or- In the Viewing window, right-click geometry within the subassembly and select Resolve All.
Resolve a part in the Viewing window	Press Shift and then, in the Viewing window, click the part. The viewer automatically determines the minimum set of JtSimplification data to resolve in order to fully load the part.

Locating assembly parts that failed to load

Overview of locating assembly parts that failed to load

When you open a 3D model, the parts that make up the assembly are listed in the **Assembly** view. You can find node parts that fail to load by:

- A node marked with a red X.
- Load failure messages.

Note:

These messages are displayed only for files that failed to load, not nodes without geometry.

- Failure summary reports.

Resolve part load failures

1. In the **Assembly** view, right-click in a clear white space and select **Retry Failed**.
2. In the **File Usage Confirmation** dialog box, click **Browse** and locate the missing part.
3. Click **Use Selected Location**.
4. Repeat the above steps for each missing part.

Tip:

Add the location of each missing part to the User-defined Part Loading Directory Set for the application to find them automatically the next time you open the model.

Turn on or off automatic load failure messages

1. Choose **File**→**Preferences**→**3D Loader**.
2. In the **3D Loader preferences** dialog box, on the **Other** tab, select or deselect **Show Load Failure Dialogs**.

Generate a summary report

1. Choose **Tools**→**Assembly**.
2. Choose **Assembly**→**Failure Report**.
3. In the **Failure Report** dialog box, choose what to include in your report:

Use this option	To
Include Load Failures	Reference JT files that failed to load with red X items in the Assembly view.
Include Invalid/No Geometry References	Reference invalid files for the node with shaded items in the Assembly view.

Tip:

You can control the type of information that is displayed in your reports, and you can disable the Reports Wizard by using the **Failure Preferences** dialog box.

4. Click **Create Report**.

Specify failure preferences

1. Choose **Assembly→Failure Preferences**.
2. Select one or more of the following parts to include in your report:
 - **Include Load Failures**
 - **Include Invalid/No Geometry References**
 - **Show wizard before report generation**

Creating part reports

Overview of creating part reports

A part report is a document that presents data in a table format, about parts in the 3D model that you are currently working with. You typically generate part reports about a part or group of parts, to examine attribute information such as angles, chordal lengths, part names, and more.

Example:

You need to figure out how much paint is needed to paint a new car. Select all the parts that will be painted, generate a part report of the parts, and use the information in the surface area attributes of the parts to calculate the amount of paint needed.

Several of the output formats of a part report (.html, .csv, .txt, .xml) make it convenient to attach a part report to an email for quick communication with co-workers, or to post on a shared Web site.

At a high level, when you create a part report, you indicate which parts of the model the part report is based on in the **Part Report-Part Set Selection** dialog box. After you make those selections, the **Report Generator Wizard** appears, and you can determine what heading and attribute information to include in the part report, and the output format of the report.

Create part reports

Use HTML part reports to list specific attributes for parts of your model.

1. Choose **Actions**→**Part Report**.
2. In the **Part Report-Part Set Selection** dialog box, select from the following:

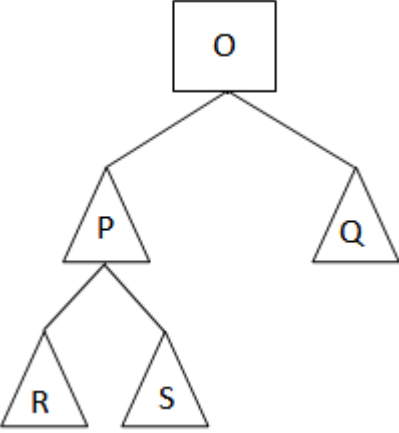
Choose this option	To do this
Visible	Create a report that includes information for all visible parts.
Selected	Create a report that includes information for selected parts.
With text markups	Create a report that includes information for only those parts that have anchored text markups. Select Include Identifier to use the text in the anchored text markup as the part identification.
Parts (Always On)	Perform analysis on all parts in the model. Note: You cannot disable part-based analysis. However, you can specify for leaf components and end items to override parts.
Leaf Component	Perform analysis on leaf components in the model.
End Item	Perform analysis on end items in the model.

3. Click **Create Report**.
4. In the **Report Definition Wizard** dialog box, follow the remaining steps.

Overview of components in Teamcenter lifecycle visualization

In Teamcenter lifecycle visualization, the **components of a 3D model** are identified based on how the model was translated from the CAD file to JT files or PLM XML files.

Component	Explanation
Part components	A part component is always the lowest level of components in the model.
Leaf components	A leaf component is a component in Lifecycle Visualization which does not have component children. Note: A sub-solid is not a CAD component, and is not considered a child of a part or assembly component in Lifecycle Visualization.

Component	Explanation
	Example: • If the model does not contain any sub-solids, the part component level of the model is the leaf component. • If the model does contain sub-solids, the lowest level of the tree that is not a sub-solid is the leaf component. A model can contain multiple levels of sub-solids. In the diagram below, O is the leaf component, as it is the lowest level in the tree that is not a sub-solid level. 
End items	End items are components with the metadata attribute <code>__PLM_END_ITEM</code>. A model may or may not have any end items defined in it, depending on how the CAD file groups the model components when the PLM XML file is created. A JT file can contain end items if the JT file was exported from a PLM XML file that had end item metadata. An end item is an assembly component that is expected to be treated as a single unit in the model—details about the parts or assembly components inside the end item are not included in the part report. Example: You have commissioned a supplier company to provide a part for a product you are building. The part report you are generating relates to the materials needed to build the product. If the assembly for the supplier company's part is marked as an end item, information about the contents of that assembly is not collected when you generate your part report. There is currently no icon in the hierarchy that identifies an end-item.

Examples of leaf components in a hierarchy

A **leaf component** is a component in Teamcenter lifecycle visualization which does not have component children.

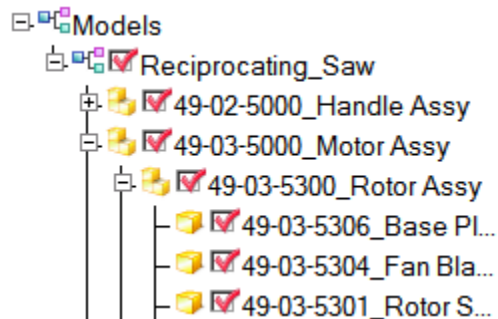
Note:

A sub-solid cannot be a leaf component.

In the hierarchy tree, sub-solids are represented by the sub-solid icon . The parent level for a sub-solid in the hierarchy is represented by the part component icon . An assembly component is a component which has other components as children. Assembly components are represented by the assembly icon .

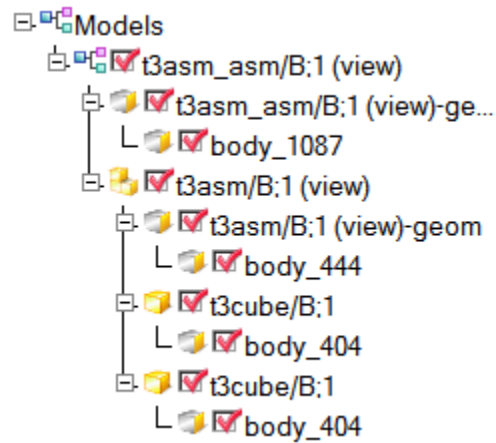
Leaf components in a model without sub-solids

In the following example model, there are no sub-solids. In this case, parts are the lowest level of the model; therefore, parts components  are the leaf components in this hierarchy.

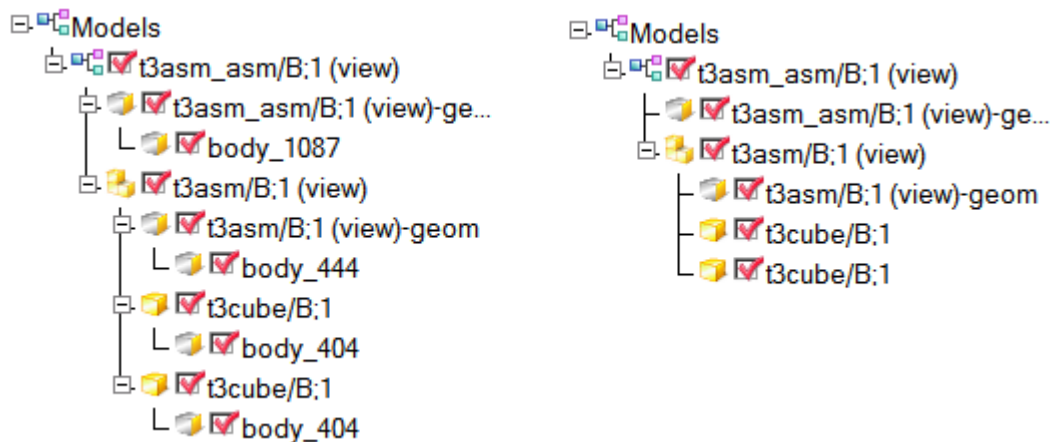


Leaf components in a model with sub-solids

In the following example model, sub-solids exist for part components and assembly components. In this case, the lowest levels of the tree which are not sub-solids are the part components ; therefore, part components are the leaf components in this hierarchy.

**Note:**

Sub-solids are still included in a part report even if they are not displayed in the hierarchy. If you do not see any sub-solids in your hierarchy, verify that the option to show leaf structure in the hierarchy is enabled. Below is an example of the same hierarchy with sub-solids showing, and with them not showing.



Overview of components in CAD models

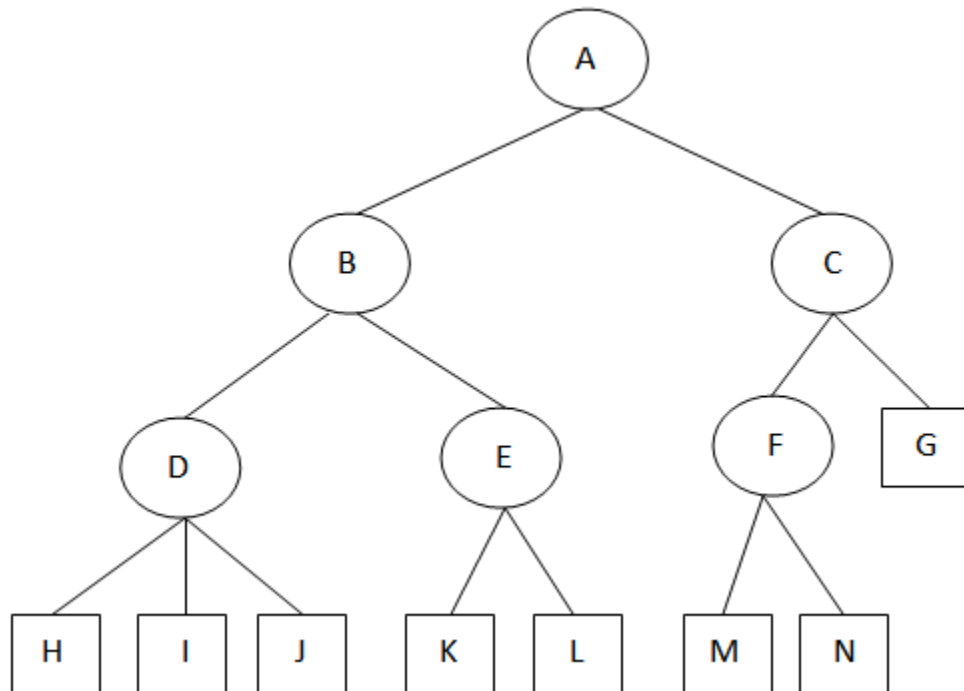
When creating part reports, it is important to understand the way assemblies, parts, and sub-solids are organized in a CAD model. An understanding of these components will help you generate a part report that contains the data you are expecting.

At an intuitive level, a 3D model is comprised of a lot of individual parts, which are usually grouped together in clusters called assemblies. Assemblies can be further grouped into larger assemblies.

Example:

In the example diagram below, the squares represent parts, and circles represent assemblies.

- Assembly D consists of parts H, I, and J. Assembly E consists of parts K and L. Assembly F consists of part M and N.
- Assembly B consists of assemblies D and E. Assembly C consists of assembly F and part G.
- Assembly A consists of assemblies B and C.



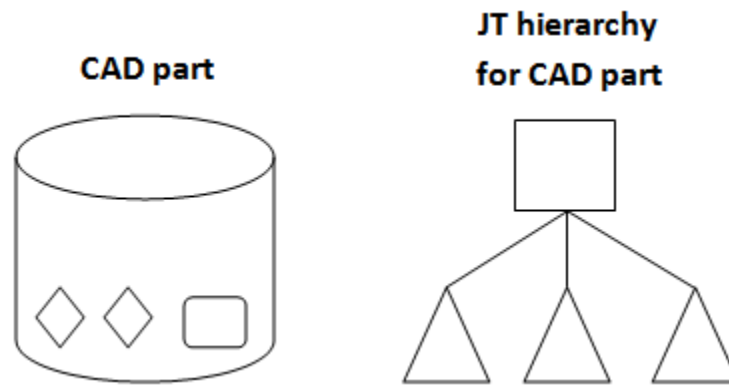
The pieces of a CAD model are referred to as part components and assembly components.

CAD components may be comprised of one or more sub-solids. A sub-solid is a piece of geometry used in the physical representation of the component. Each sub-solid object can be individually associated with a specific layer or reference set by the CAD system.

Example:

In the example diagrams below, a part component in a CAD model (represented by a cylinder) consists of three pieces of sub-solid geometry.

When the CAD model is translated to JT files, the sub-solids for the CAD part component are defined as sub-solids (represented as triangles) of the JT part (represented as a square).



Note:

A sub-solid is not a component in a CAD model.

Viewing NX advanced assembly operations

Teamcenter lifecycle visualization supports many of the advanced geometry concepts offered by NX, including arrangements, reference sets, assembly-level geometry, and promotions.

Note:

NX advanced assembly operations are supported in JT data saved from NX to the Teamcenter managed environment. Although some of these features are partially supported in JT data saved outside of Teamcenter to a local or network file system, the targeted workflow is NX→Teamcenter→visualization clients.

Arrangements

NX arrangements are collections of occurrence transformations or suppressions within an assembly. Arrangements may also include Variable Component Positioning (VCP), which is the ability to override the position of a specific instance or occurrence of a component or subassembly in the context of a parent assembly. Lifecycle Visualization displays the last saved or active arrangement from NX or the Structure Manager.

Note:

Support for arrangements has the following limitations:

- The data model may be accurate only for the last saved or active arrangement in NX.
- Lifecycle Visualization displays only a single arrangement, which is specified in Teamcenter before the model is opened in the viewer.

- The viewer supports arrangements only in PLM XML-based models from the Teamcenter managed environment.
- JT data exported from Lifecycle Visualization does not include arrangements. However, arrangements are preserved in exported PLM XML.
- To use Massive Model Visualization (MMV) with arrangements, create the **PSEIgnoreArrangements** preference in Teamcenter if it does not exist and set it to **True**.

Reference sets

NX reference sets are named configurations of geometry applied in the context of an assembly. A single component JT file contains all of the information necessary to define the geometry configurations for each reference set.

The loaded reference set determines what geometry is active for a given component. This geometry is used for the purposes of visualization of the component, as well as all analysis operations. Component bodies that do not belong to the loaded reference set are grayed out in the assembly tree

Note:

Support for reference sets has the following limitations:

- Product structure changes are not supported via reference sets. The closest approximation of this behavior within NX is when the **Apply to All Assembly Levels** reference set load option is turned on for a reference set.
- Support of the **Empty** reference set within Lifecycle Visualization is limited and may not exactly reproduce its treatment in NX.
- Reference set transforms are not supported.
- Component exclusion via reference set is not supported.
- PMI references via reference set are not supported.
- If a reference set contradicts an applied layer filter for any given part, the reference set is not applied to the part. Likewise, if a layer filter contradicts an applied reference set for a part, the layer filter is not applied to the part.

Assembly-level geometry

Assembly-level geometry may consist of model geometry, reference geometry, wireframe data, and points. Both standalone and embedded viewers support assembly-level geometry via JT files vaulted at the assembly-level in Teamcenter.

Promotions

NX promotions consist of component geometry copied into the owning assembly and then modified. Although the component geometry is overridden when it is displayed in the context of the assembly, the geometry within the component itself is unchanged.

Both standalone and embedded viewers support promotions via JT datasets that are created in Teamcenter at the assembly-level; these JT datasets are attached to absolute occurrence overrides of the component for which the geometry is overridden.

Note:

Support for promotions has the following limitations:

- The data model may be accurate only for the last saved or active arrangement in NX.
- Geometry may become out of sync due to the dependency of one component upon another.
- There is only limited support for promotions outside of the Teamcenter managed environment. If an assembly and its subassembly contain promotions and the overall assembly is translated from NX to JT, then only the visualization of the overall assembly is consistent with the original data in NX. When visualized separately, the subassembly is not consistent with NX.

View NX part name and count information

You can view the name that is associated with a part in NX, as well as the number of components that correspond to the NX assembly, from the **Assembly** view. The component count corresponds to the number of lines in the Bill of Materials for the selected item.

In the **Assembly** view, right-click the assembly, and click **Properties**.

The part name and component count appear in the **Properties** dialog box.

Compare different versions of your assembly

You can save your assembly to a session file, and then compare it with another version of the assembly. If the updated assembly contains additions or deletions, the changes are noted in the **Assembly** view.

1. Open a file that contains an assembly.
2. Choose **Tools**→**Assembly**→**Reconcile Hierarchy**.
3. Save and then close a session file.

The entire assembly is written to the session file.

4. In your CAD package, update your assembly file, and save it to the same location referenced by the session file, using the same name.

Note:

Reconcile Hierarchy detects additions to and deletions from an assembly. It does not detect changes to existing parts.

5. Choose **Assembly**, and make sure that you have selected **Reconcile Hierarchy**.

Note:

Reconcile Hierarchy remains selected until you clear it, even if you close the application.

6. Open the Session file.

The assembly is displayed in the **Assembly** view.

7. In the **Assembly** view, right-click a column heading, and select **Rec. Hierarchy Result**.

The **Rec. Hierarchy Result** column appears. Assembly changes are marked in this column as **New** or **Deleted**. Assembly deletions are also identified by an icon, .

Export node and model names

You can save the parts and parent assemblies to a text file by exporting node and model names, and then run a custom executable using those names as input.

1. (Optional) To export the names for selected parts, select the parts.
2. To export the names for a single part, right-click the part. Otherwise, right-click anywhere in the Viewing window, and then click **Export Attribute List**.
3. In the **Export Attribute Query** dialog box, select an option from the **Export Attributes For** section.

Select this option	To export names for
Visible Parts	All visible parts.
Selected Parts	Selected parts.
Single Part	The part on which you selected.

4. In the **Define Attribute** section, select the names to include (**Node Name** or **CAD Model Name**).

If you select both options, the appropriate model name will be listed with each node name.

Note:

CAD model names are included only if the model name is saved in the model data.

5. Export the names in one of these ways:

- To save the names as a text file, click **Save to File**.

The names are saved with a **.txt** extension.

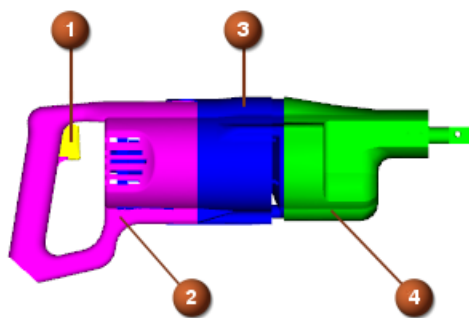
- To run a custom executable using a file containing the names as input, click **Execute Macro**.

Part groups

Part groups overview

You can create named groups of parts and do any of the following:

- Select all the parts in a group by clicking the group name in the **Groups** view.
- Display or hide all the parts in a group by selecting or clearing its check box.
- Change the appearance of all parts in a group.
- Create an alternate hierarchy from a group.
- Use part groups to perform group clearance analysis. A group clearance analysis checks for clearance violations between groups, but does not check clearances between parts within each group.



- 1 Electrical Assembly Group
- 2 Handle Assembly Group
- 3 Motor Assembly Group
- 4 Gearbox Assembly Group

The part group names appear in the **Groups** view.

If you open a model and insert one or more other models, you can group parts from multiple models. Parts can be members of more than one group. In addition, part groups are saved in session files.

Create, sort, and reorder part groups

There are two ways to specify parts for a group. You can add all visible parts or add all selected parts.

Add all visible parts to a new group

1. In the Viewing window or the assembly tree, turn on only the parts you want in the group. (Turn off all other parts.)
2. If it is not already open, show the **Groups** view.
3. Do one of the following:
 - In the **Groups** view, right-click and choose **Add Visible**.
 - Choose **Tools→Groups**, and then **Groups→Add Visible**.
4. (Optional) Rename the new group:
 - a. Right-click the group, and choose **Rename**.
 - b. Type the new name.
5. (Optional) Change the order of groups in one of these ways:
 - Right-click anywhere in the **Groups** view, and choose **Sort→Ascending** or **Descending**.
 - Drag a group to move it to a different location in the list. To move multiple groups, hold Shift or Ctrl as you select the groups.

Add selected parts to a new group

1. In the Viewing window or the assembly tree, select the parts you want in the group.
2. If it is not already open, show the **Groups** view.
3. Do one of the following:
 - If it is not already open, show the **Groups** view.
 - Choose **Tools→Groups**, and then **Groups→Add Selected**.

4. (Optional) Rename the new group:
 - a. Right-click the group, and choose **Rename**.
 - b. Type the new name.
5. (Optional) Change the order of groups in one of these ways:
 - Right-click anywhere in the **Groups** view, and choose **Sort→Ascending** or **Descending**.
 - Drag a group to move it to a different location in the list. To move multiple groups, hold Shift or Ctrl as you select the groups.

Create alternate hierarchies from groups

1. In the **Groups** view, select the group name.
2. Do one of the following:
 - Right-click the group name and choose **Copy to Alt. Hierarchy**.
 - Choose **Tools→Groups**, and then **Groups→Copy to Alt. Hierarchy**.

Display, hide, and select part groups

1. If it is not already open, show the **Groups** view.
2. Do any of the following:

To	Do this
Select all parts in a group	Click the group name.
Display parts in a group	a. (Optional) Do one of the following to set Single Group Mode status: • In the Groups view, right-click and select or clear Single Group Mode. • Choose Groups→Single Group Mode. b. Select check box to the left of the group name. Single Group Mode affects which parts appear.

To	Do this
	- With Single Group Mode off, selecting the check box next to the group name displays all the parts in the group, but does hide any other parts. If you have multiple groups, you can select more than one check box to display more than one group at a time. - With Single Group Mode on, selecting the check box next to the group name displays all the parts in the group and hides all other parts. You can only select one check box at a time so only one group displays at a time.
Hide parts in a group	a. (Optional) Do one of the following to set Full Subtract status: - On the Groups page, right-click and select or clear Full Subtract. - Choose Groups→Full Subtract. b. Clear check box to the left of the group name. If you have multiple groups displayed, Full Subtract affects which parts are hidden. - With Full Subtract on, hiding group A hides all parts in group A, even if some parts are also in group B. - With Full Subtract off, hiding group A does not hide group A parts that are also in group B.

Note:

You can use the **Undo** command to undo the last selection or visibility action.

Replace parts in a group

There are two ways to replace parts in a group. You can replace the parts with all visible parts or all selected parts.

Replace parts in a group with all visible parts

1. In the Viewing window or the assembly tree, turn on only the parts you want in the group. (Turn off all other parts.)
2. If it is not already open, show the **Groups** view.
3. Do one of the following:

- In the **Groups** view, right-click and choose **Replace Visible**.
- Choose **Tools→Groups**, and then **Groups→Replace Visible**.

Replace parts in a group with selected parts

1. In the Viewing window or the assembly tree, select the parts you want in the group.
2. If it is not already open, show the **Groups** view.
3. Do one of the following:
 - In the **Groups** view, right-click and choose **Replace Selected**.
 - Choose **Tools→Groups**, and then **Groups→Replace Selected**.

Change the appearance of parts in a group

1. If it is not already open, show the **Groups** view.
2. Do one of the following:
 - In the **Groups** view, right-click and select **Use Group Appearance**.
 - Choose **Tools→Groups**, and then **Groups→Use Group Appearance**.

Note:

If **Use Group Appearance** is not selected, you will not see group appearances.

3. In the **Groups** view, right-click the group name and choose **Edit Group Appearance**.
4. In **Group Appearance Editor** dialog box, do any of the following:

To	Do this
Change color	Click a color button and then choose a color from the color well or create a custom color. You can specify ambient, diffuse, specular, and emissive colors.
Change transparency	Enter a Transparency value between 0 and 1.

To	Do this
	Note: A part with an transparency value of 1 is invisible; a part with an transparency value of 0 is opaque.
Change shininess	Enter a Shininess value between 0 and 128. Note: A part with a shininess value of 0 is very shiny; a part with a shininess value of 128 is dull.
Preview the changes	Click Apply .
Cancel the changes	Click Cancel .

Note:

You can use the **Undo** command to undo the last **Group Appearance Editor** action.

Return groups to their original appearance

Clear **Use Group Appearance** in one of these ways:

- In the **Groups** view, right-click and clear **Use Group Appearance**.
- Choose **Tools**→**Groups**, and then **Groups**→**Use Group Appearance**.

Filtering parts

Using filters to load, display, or select parts

Filters find parts that match criteria that you define, and then load, display, or select the parts. You can apply filters in series, so that only parts that pass the first filter are considered for the second filter, and only parts that pass the second filter are considered for the third filter.

The following are examples of how you can use filters.

- For large assemblies, create a volume filter to load only the parts that are in a small area of the assembly.
- Create a name filter to hide all parts with part names that start with *elec*.

- If your assembly model includes part material metadata, create an attribute filter to select all parts made of aluminum.
- Create a Clearance filter to show only parts that are in contact with a selected part.
- Apply two filters in series to select all aluminum parts that are in contact with a target part.

Create filters to load, display, or select parts

1. Choose **Actions**→**Filters**.
2. From the **Available Filters** list in the Filter Manager, select a **filter type** and click **Add**.

The filter is added to the **Filter Sequence** list. This list determines the order in which filters are applied.

3. From the **Filter Sequence** list, select the filter.
4. From the **Filter Input Select** list, specify which parts to filter:

To filter	Select
All parts in the assembly, whether or not they are loaded	All Parts
Parts that are loaded, whether or not they are displayed	Loaded Parts
Parts that are displayed	Visible Parts
Parts that are selected	Selected Parts
Parts that pass the previous filter in the Filter Sequence list	Previous Output

5. From the **Filter Output Action** list, specify what happens to the parts that match the filter criteria.

To	Select
Make no changes to the part state as a result of the filter	None
Display only the parts that pass the filter and hide all parts that do not pass the filter	Show Only
Display the parts that pass the filter, in addition to parts that are currently displayed	Show Additive
Hide the parts that pass the filter	Hide
Select the parts that pass the filter and deselect all parts that do not pass the filter	Select Only

To	Select
Select the parts that pass the filter, in addition to parts that are currently selected	Select Additive
Deselect the parts that pass the filter	Deselect

6. Click **Properties** and **set the filter properties**.
7. (Optional) To rename the filter, right-click the filter and choose **Rename**.
8. Select the check boxes only for filters that you want to apply.
9. Click **Apply**.

The filter output action is applied.

Note:

If you create a **Name** or **Attributes** filter, and you filter by parts and assemblies, the filter output action is applied only to the topmost matching node of each assembly, even if it has child parts that fit the criteria.

10. (Optional) **Save** filters that you want to use later.

Note:

Unsaved filters are lost when you close the Filter Manager.

11. (Optional) Export saved filters to a file.

Filter types

Overview of filter types

This type	Examines
Attributes Filter	Metadata attached to parts and assemblies.
B-Rep Filter	Part B-Rep data.
Clearance Filter	Parts within a specified distance of target parts.
Compound Filter	A series of other filters.
Irregular Zone Filter	Parts that are inside, outside, or intersecting an irregularly shaped volume you define.

This type	Examines
Name Filter	Part and assembly names.
PMI Filter	Parts with PMI.
Visibility Filter	Parts that are visible from specified viewpoints.
Volume Filter	Parts that are inside or outside a box or sphere you define.

Set attribute filter properties

Overview of attribute filters and searching for attributes

Parts can have many attribute. Some attributes are defined in the assembly and model files. Models in Teamcenter have additional Teamcenter attributes.

When you set filter properties for an attribute filter, you define the number and kind of attributes to display and work with in the Viewing window. You configure attribute filters to search for attribute properties in the following ways:

- You configure attribute filters to search for attribute properties that are saved in the local file system. This configuration is also called a cache search because all targeted parts and assemblies must be opened and available for searching (local cache). If the assembly contains numerous parts and subassemblies, the search may be time consuming.
- You configure attribute filters to first **search for BOM lines** in Teamcenter, and then work with attribute properties that are now saved to your local cache.
- You configure attribute filters to **search Teamcenter** for **Items, Item Revisions, Occurrence Notes, or Saved Queries**. This configuration is called a cacheless search because the application processes the attribute search on the Teamcenter server, and because the application loads those objects that match the search criteria. Search performance is expected to be better than a local cache search where all objects must be loaded before the search process begins.

Attribute filters include an **Operator**, for example, *JT:CAD Source = NX* or *Translation Date < 2022/01/01–12:30:00*. Available operators are dependent on operators that are supported by the attribute. The complete list of operators includes the following:

Attributes Filter Properties

Source:Attribute... **Operator** Value... - +

☒ Case sensitive Clear

Filter By:

☐ Parts ☐ Assemblies ☒ Parts and Assemblies

Filter Options

☒ Use local filtering ☐ Use Teamcenter filtering

OK Cancel

Symbol	Meaning
<	Less than
<=	Less than or equal to
!=	Not equal to or does not match
=	Equal to or matches the value
>	Greater than
>=	Greater than or equal to

Note:

- If you want to specify a search range, use these two operators – ">" and "<" to define the range.
- Some attributes only support the equal operation, which means the **Operator** list may only contain the = operator.

- Use the = operator for cacheless searches of Teamcenter *before* and *after* attributes, such as *Created Before* and *Created After*. For these Teamcenter attributes, *Before* implies < less than. *After* implies >= greater than or equal to. *Created Before* = 2022/01/01 finds parts created before 1 Jan 2022. *Created After* = 2022/01/01 finds parts created after 1 Jan 2022 *and* parts created on 1 Jan 2022.

You can add or remove as many attribute-value pairs as you need.

- Use the + button to add a new entry to the list. All entries listed are combined together using a logical *AND* to perform the search.
- Use the - button to remove current entry from the list. The button is unavailable if only one attribute is listed.

Attributes Filter Properties

JT:Name	=	6_Fixture	-	+
Source:Attribute...	Operatc	Value...	-	+
Source:Attribute...	Operatc	Value...	-	+
Source:Attribute...	Operatc	Value...	-	+

☒ Case sensitive Clear

Filter By:

☐ Parts ☐ Assemblies ☒ Parts and Assemblies

Filter Options

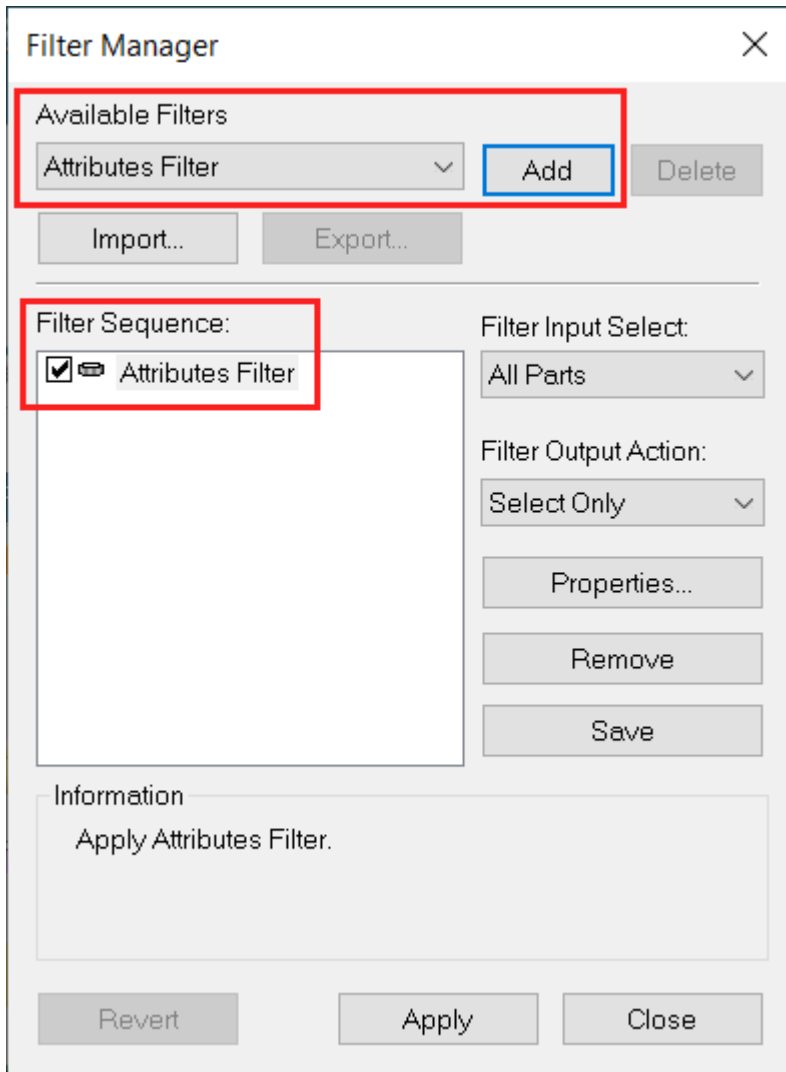
☒ Use local filtering ☐ Use Teamcenter filtering

OK Cancel

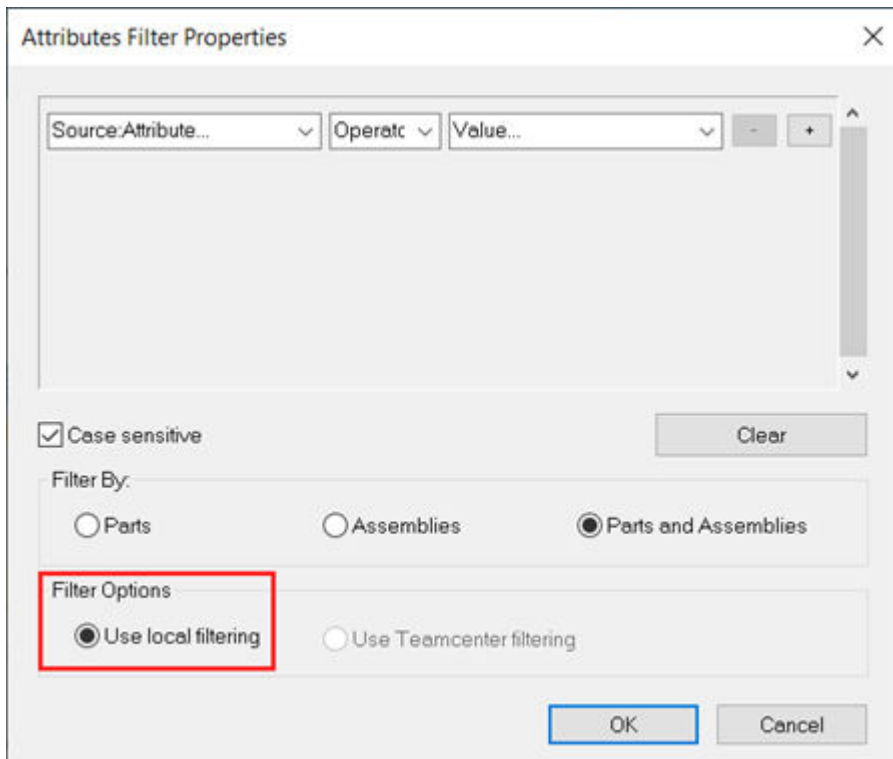
Set attribute filter properties to search the local file system

- Choose **Actions**→**Filters**.

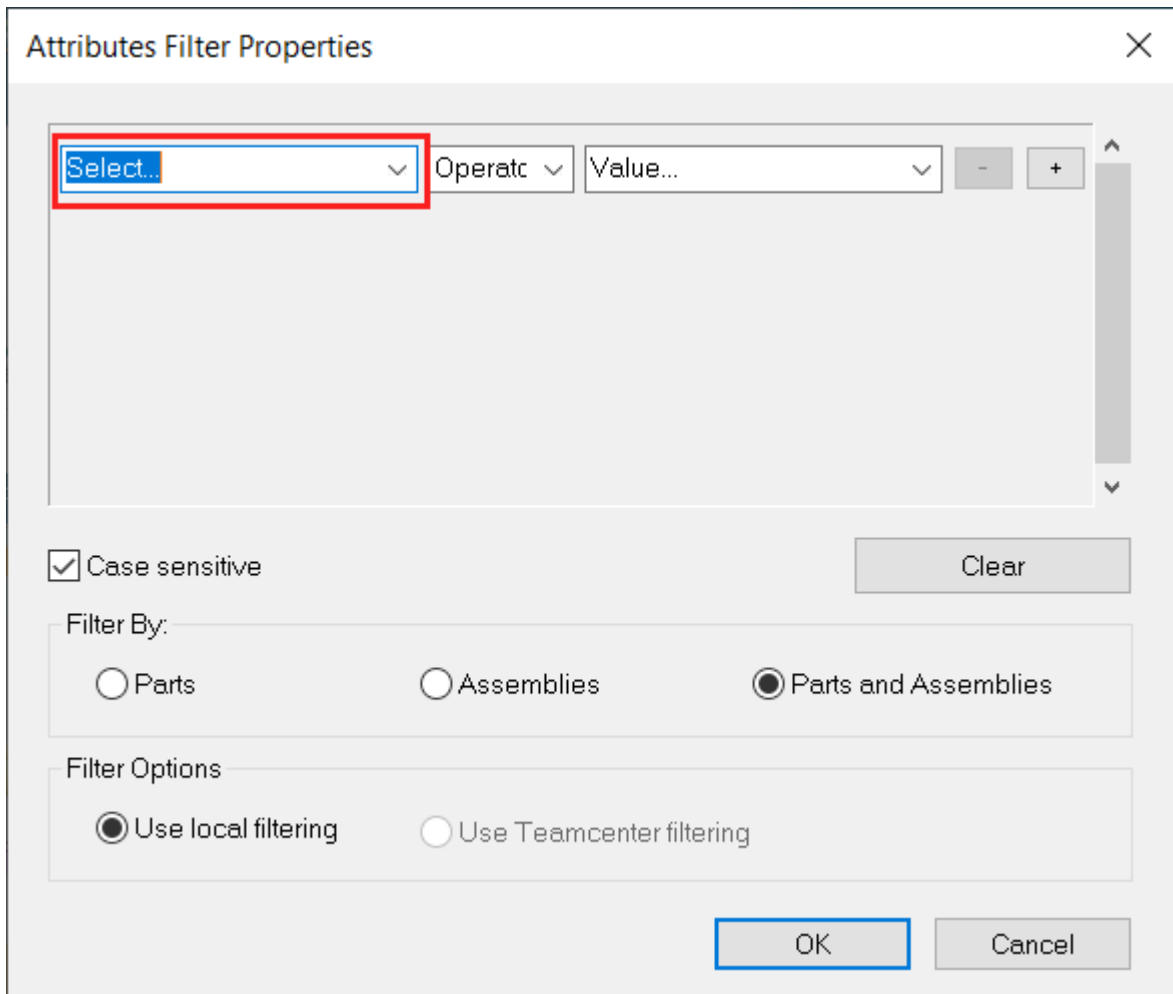
-
2. (If an attributes filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the **Attributes Filter** and click **Add**.



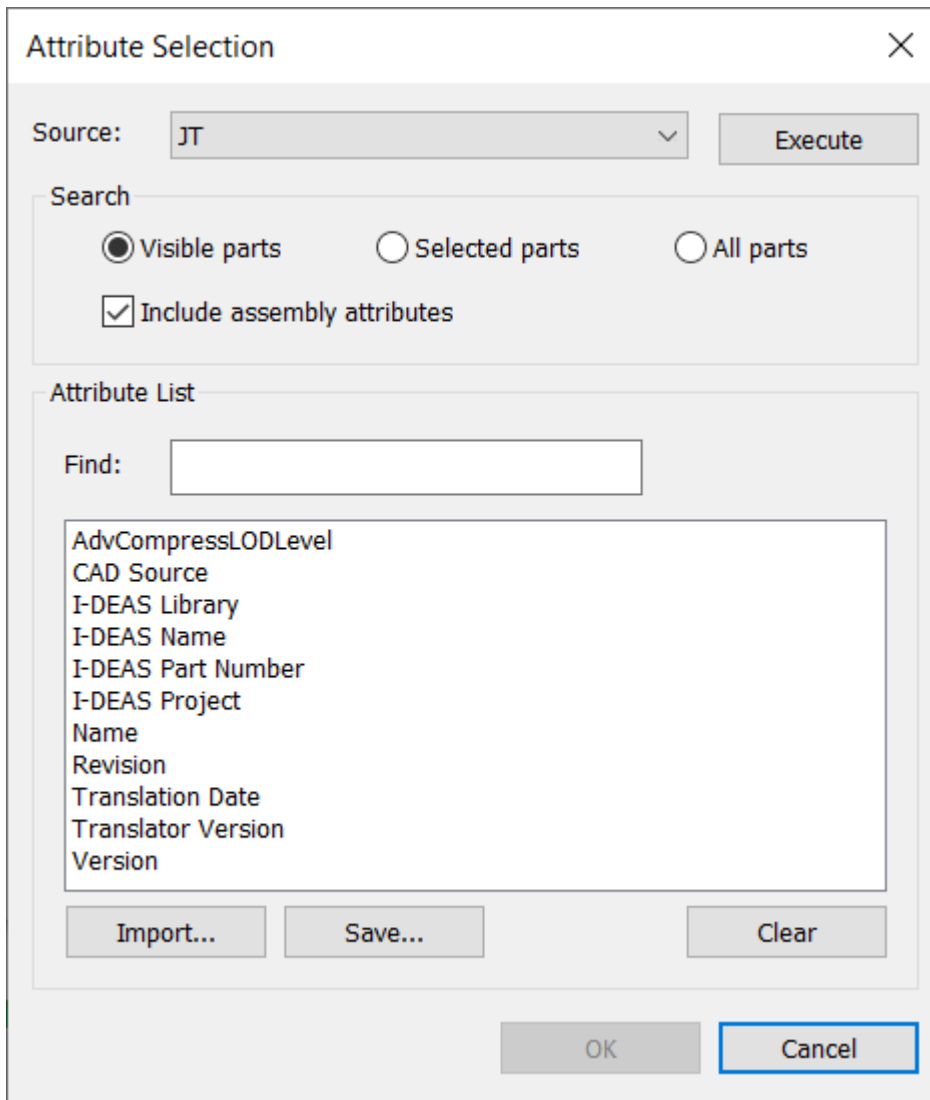
-
-
3. In the **Filter Sequence** list, select an attributes filter.
4. Click **Properties** to display the **Attributes Filter Properties** dialog box.
5. In the **Attributes Filter Properties** dialog box, in the **Filter Options** section, choose **Use local filtering**, if it is not already selected.



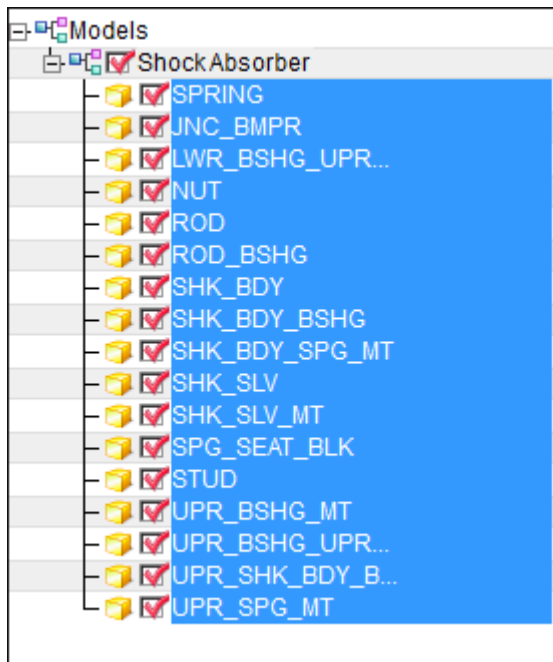
6. In the **Source:Attribute** list, choose **Select**.



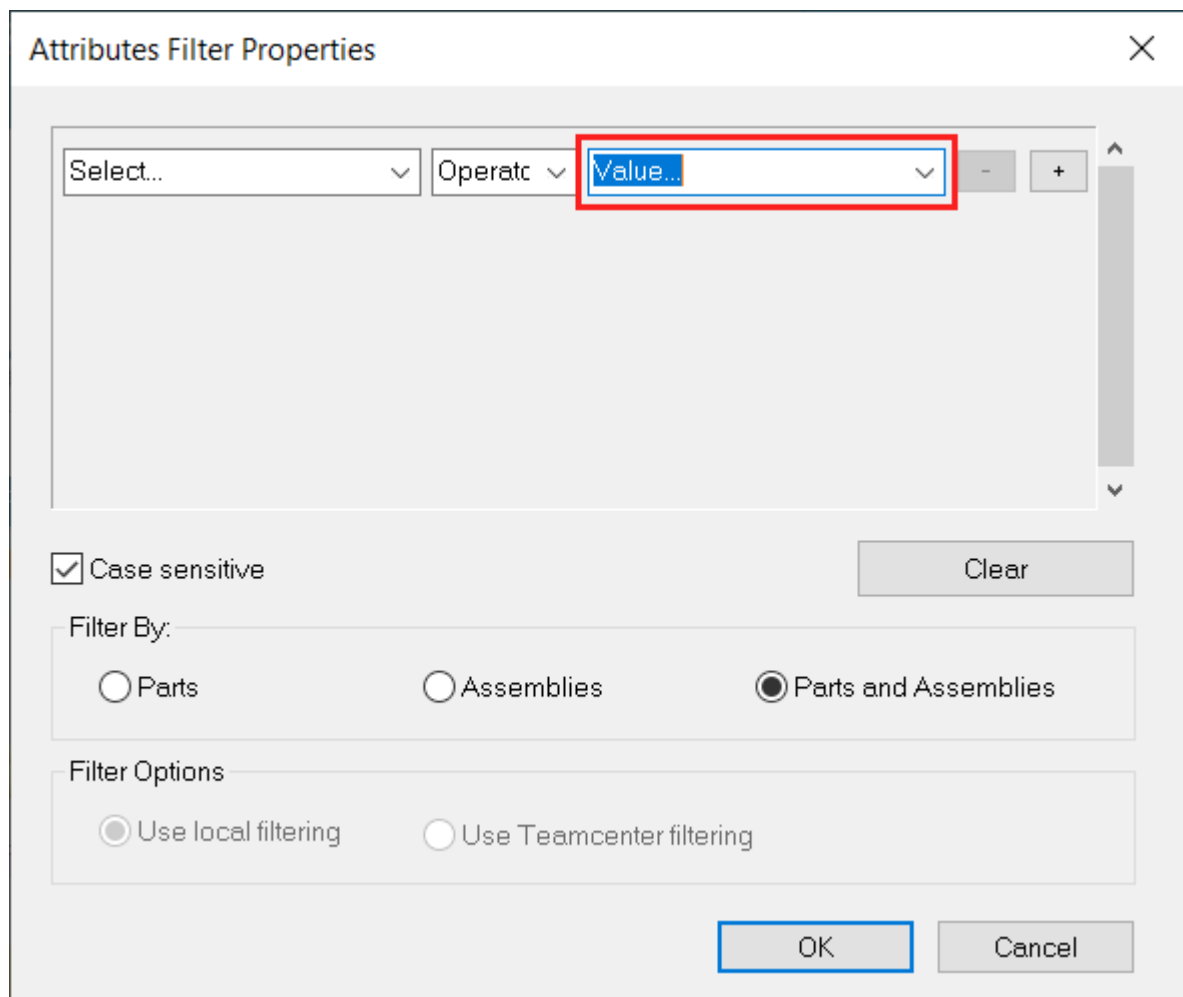
7. In the **Attribute Selection** dialog box that appears, select **JT** from the **Source** list.



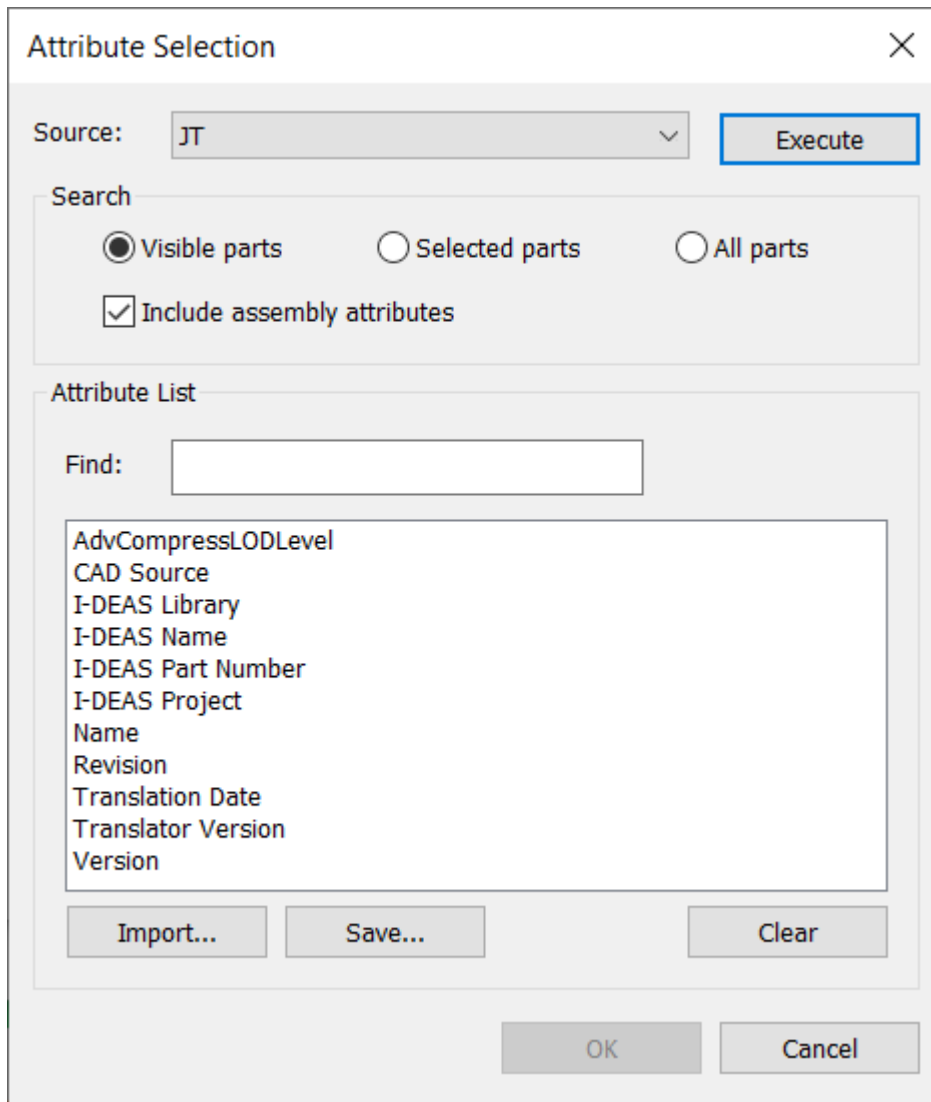
8. In the **Search** section, choose **Visible parts**, **Selected parts**, or **All parts**, and then click **Execute** to generate the list of JT attributes contained in the assembly.
9. In the **Attribute List** section, search for an attribute by typing in the **Find** section, or by highlighting the attribute in the list. When finished, click **OK**.
10. In the **Attributes Filter Properties** dialog box, follow these steps to specify an attribute value to filter:
 - a. In the Assembly tree, select one or more parts.



- b. In the **Attributes Filter Properties** dialog box, in **Value**, click **Select**.



- c. In the **Attribute Values** dialog box that appears, select a value.



The **Attribute Selection** dialog box is shown. It has a title bar with a close button (X). The **Source:** dropdown is set to **JT**, and the **Execute** button is highlighted with a blue border. Below this is a **Search** section with three radio buttons: **Visible parts** (selected), **Selected parts**, and **All parts**. There is also a checked checkbox for **Include assembly attributes**. The **Attribute List** section contains a **Find:** text box and a list of attributes: **AdvCompressLODLevel**, **CAD Source**, **I-DEAS Library**, **I-DEAS Name**, **I-DEAS Part Number**, **I-DEAS Project**, **Name**, **Revision**, **Translation Date**, **Translator Version**, and **Version**. At the bottom of the list are **Import...**, **Save...**, and **Clear** buttons. At the very bottom of the dialog are **OK** and **Cancel** buttons.

d. (Optional) In **Value**, type a known attribute value using **optional wildcards**.

11. Click **OK**.
12. In the **Filter Manager** dialog box, click **Apply**.

The **Attributes Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

13. (Optional) Click **Revert** to revert the display to its original state. You can switch the display from using the attributes filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Set attribute filter properties to search Teamcenter structure lines

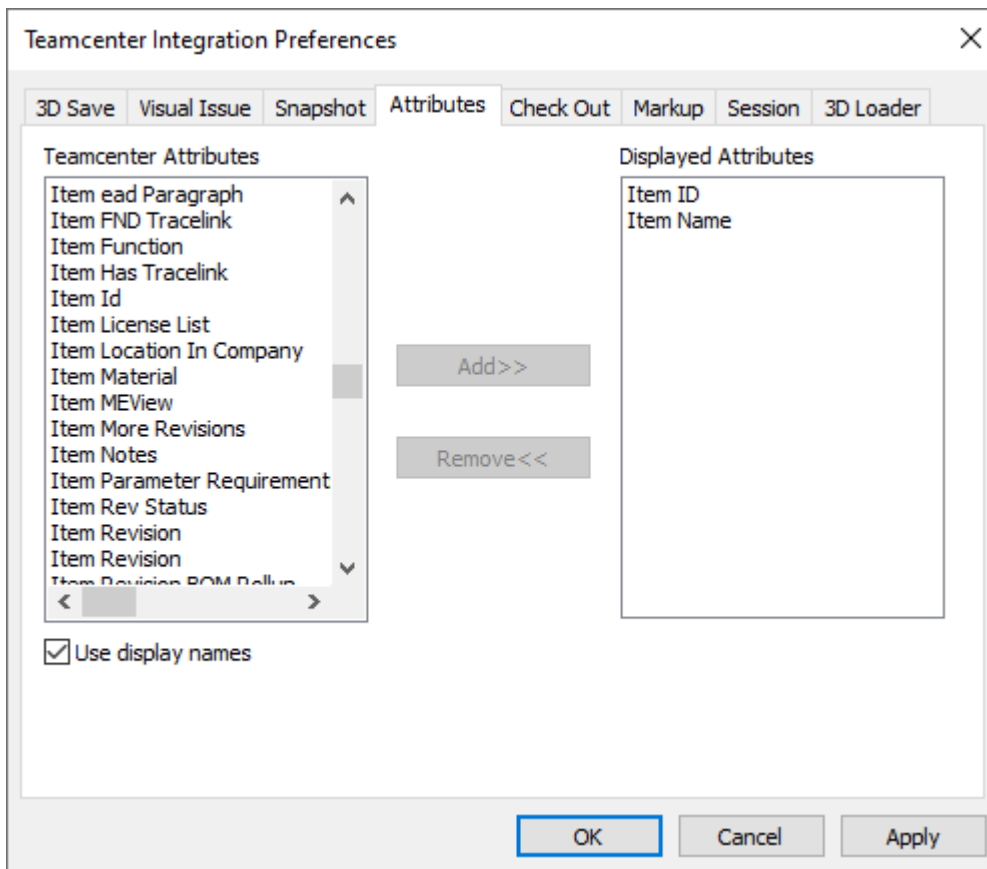
Prior to setting the attribute filter properties, you may need to specify the BOM line attributes to be loaded. If you are interested in 4G design element attributes, you do not need to specify the attributes to be loaded.

1. Choose **File→Preferences→Teamcenter Integration**, and then in the **Teamcenter Integration Preferences** dialog box, click **Attributes**.

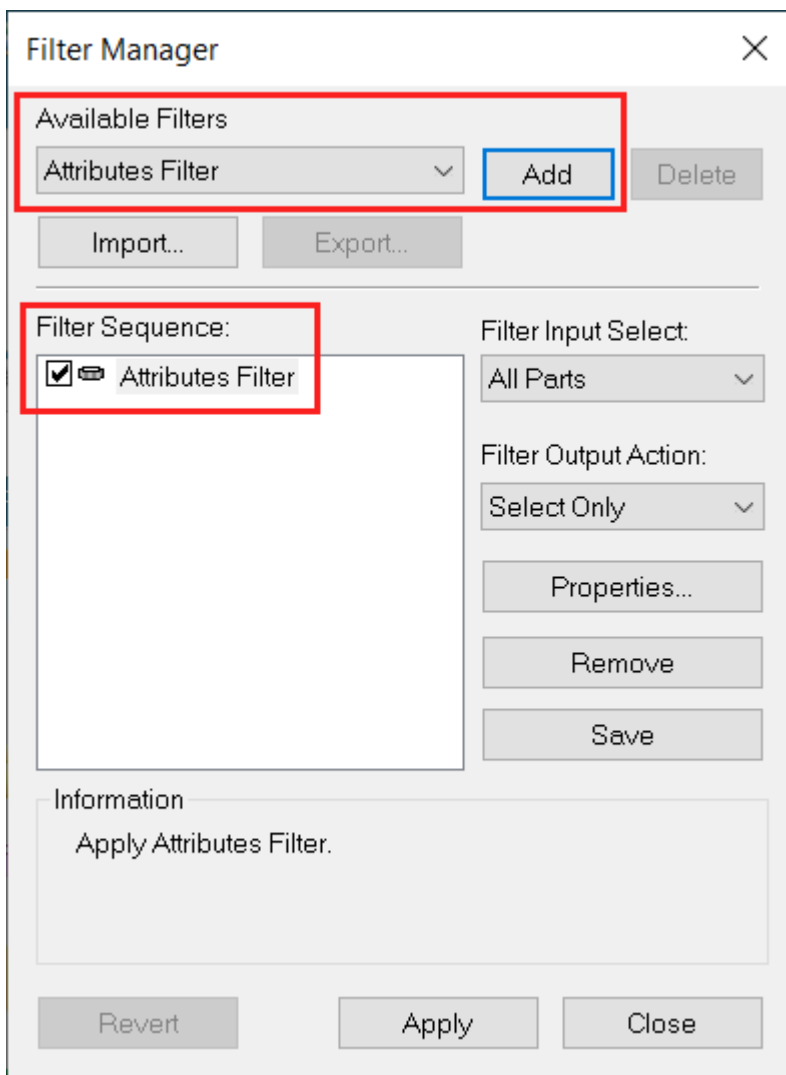
Note:

You must specify structure line attributes by loading 3D data from Teamcenter.

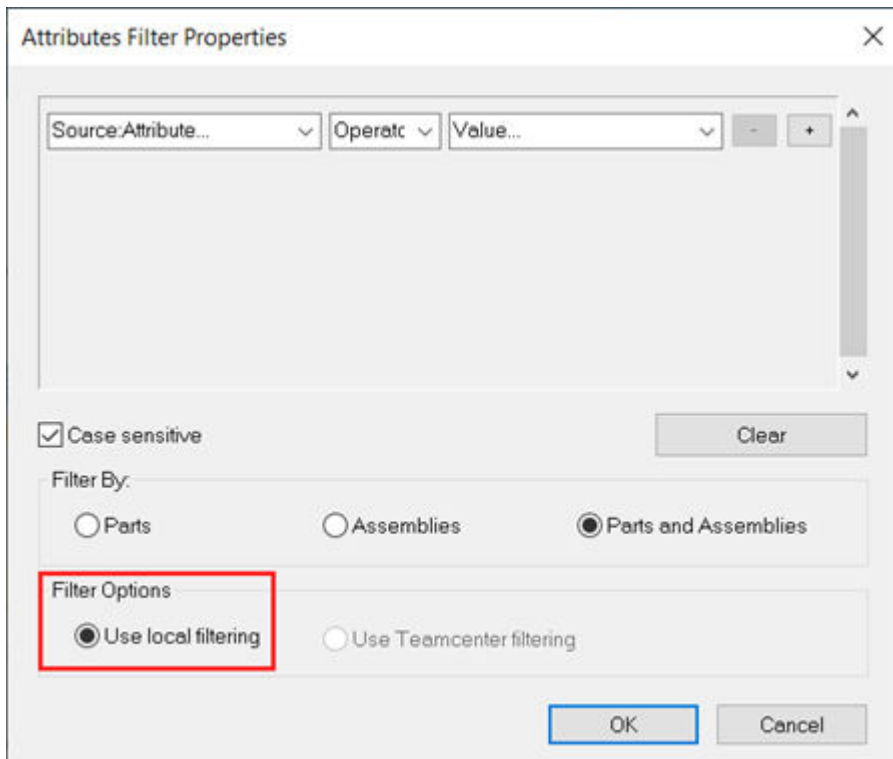
2. In the **Attributes** tab, from the **Teamcenter Attributes** list, add attributes to **Displayed Attributes**.



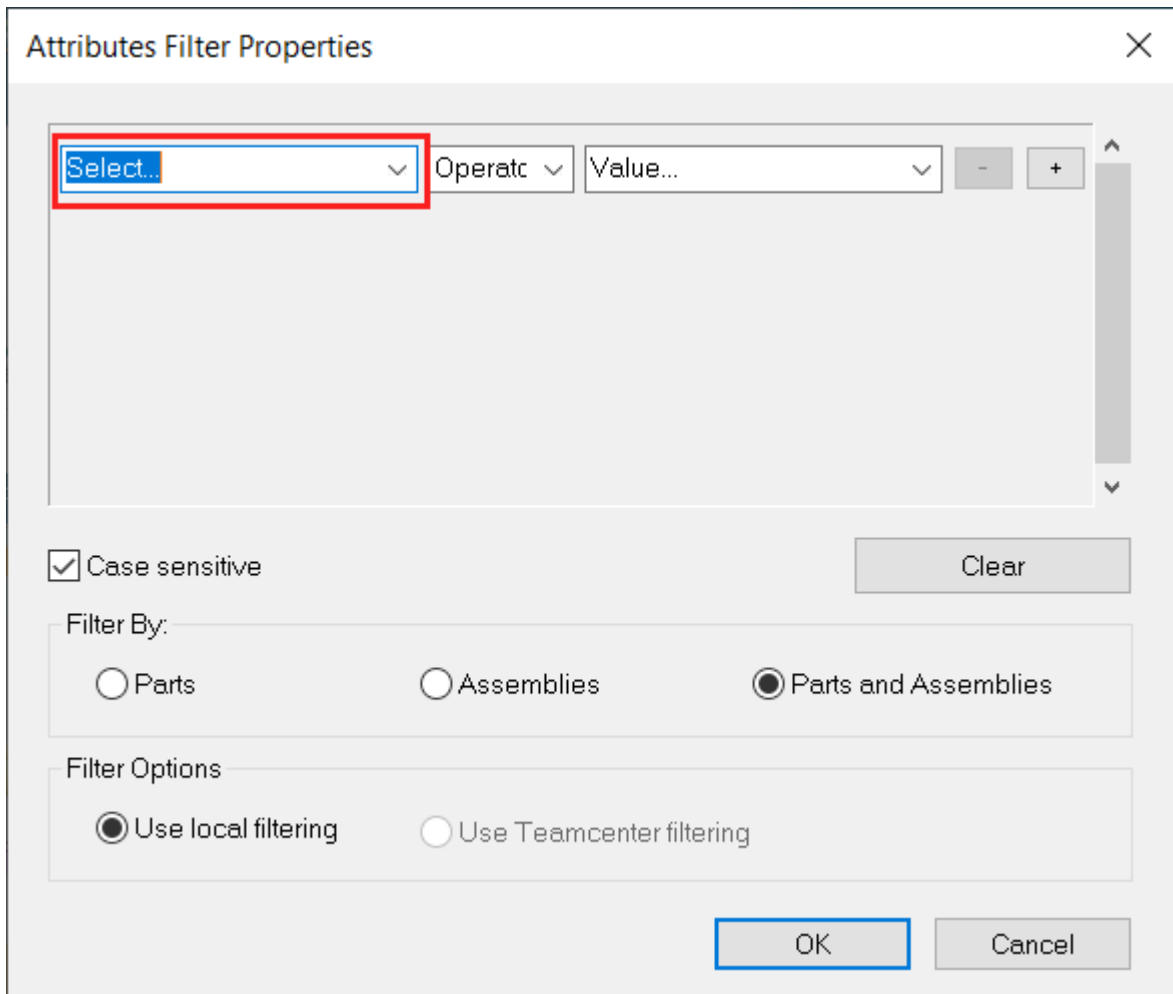
3. Close and reopen the file for the preference change to take effect.
4. Choose **Actions→Filters**.
5. (If an attributes filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the **Attributes Filter** and click **Add**.



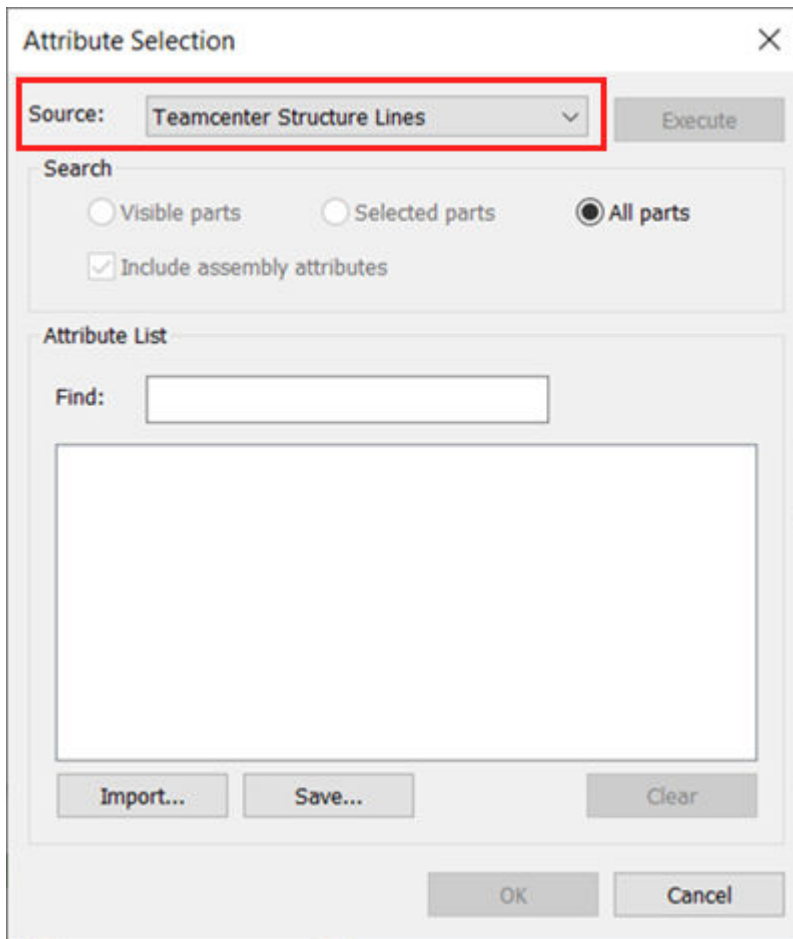
6. In the **Filter Sequence** list, select an attributes filter.
7. Click **Properties** to display the **Attributes Filter Properties** dialog box.
8. In the **Attribute Filter Properties** dialog box, in the **Filter Options** section, choose **Use local filtering**.



9. In the **Source:Attribute** list, choose **Select**.



10. In the **Attribute Selection** dialog box, select **Teamcenter Structure Lines**.



11. Do one or more of the following:

To	Do This
Generate the list of Teamcenter Structure Line attributes contained in the assembly.	- In the Search section, choose Visible parts, Selected parts, or All parts. - Click Execute to generate the list.
Search for a specific attribute. in the Attribute List section.	- In the Find field, enter an attribute name. - Select the attribute from the returned list.

12. Click **OK**.

13. Do the following to specify an attribute value to filter on.

- In the assembly tree, select one or more parts.
- In the **Attributes Filter Properties** dialog box, from the **Value** dropdown, choose **Select**.

You can also search for a known attribute value by typing the attribute value in the **Value** dropdown. Wildcard values may be used in the search.

- c. In the **Attribute Values** dialog box, highlight a value to select it.
- d. Click **OK**.

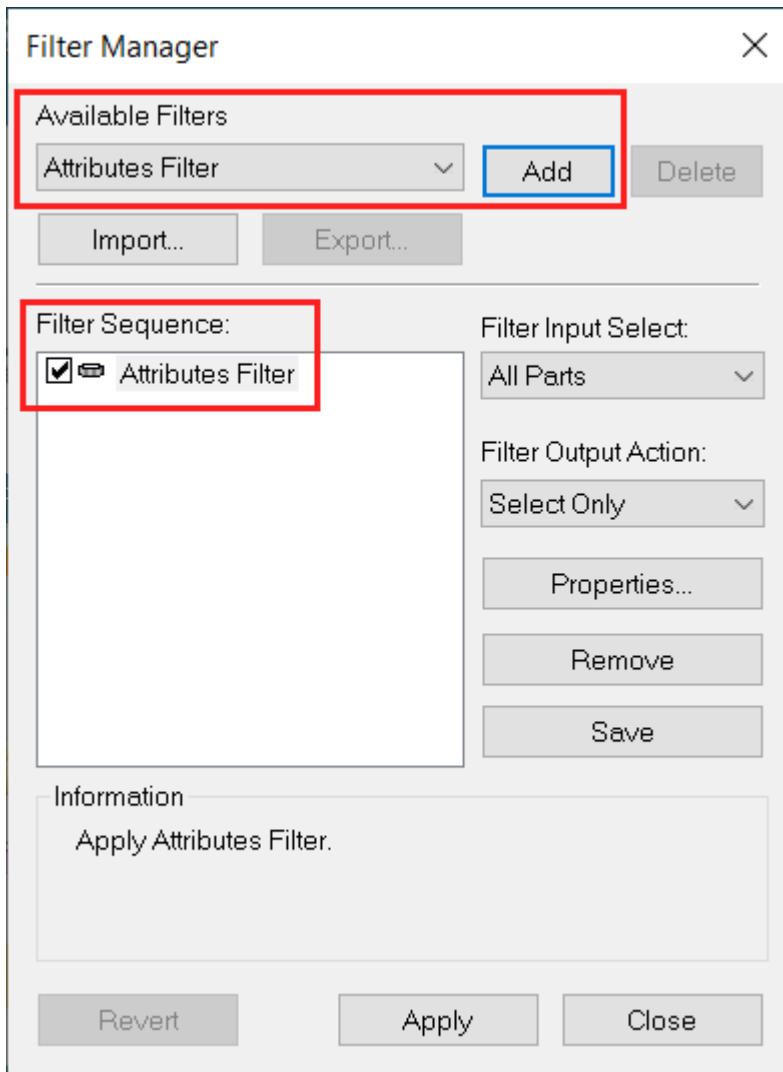
14. In the **Filter Manager**, click **Apply**.

The attribute filter properties are set. When you apply the filter, parts will be filtered based on the selected properties.

15. (Optional) Click **Revert** to revert the display to its original state. You can switch the display from using the attributes filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Set attribute filter properties to search Teamcenter using cacheless search

1. Choose **Actions→Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the **Attributes Filter** and click **Add**.



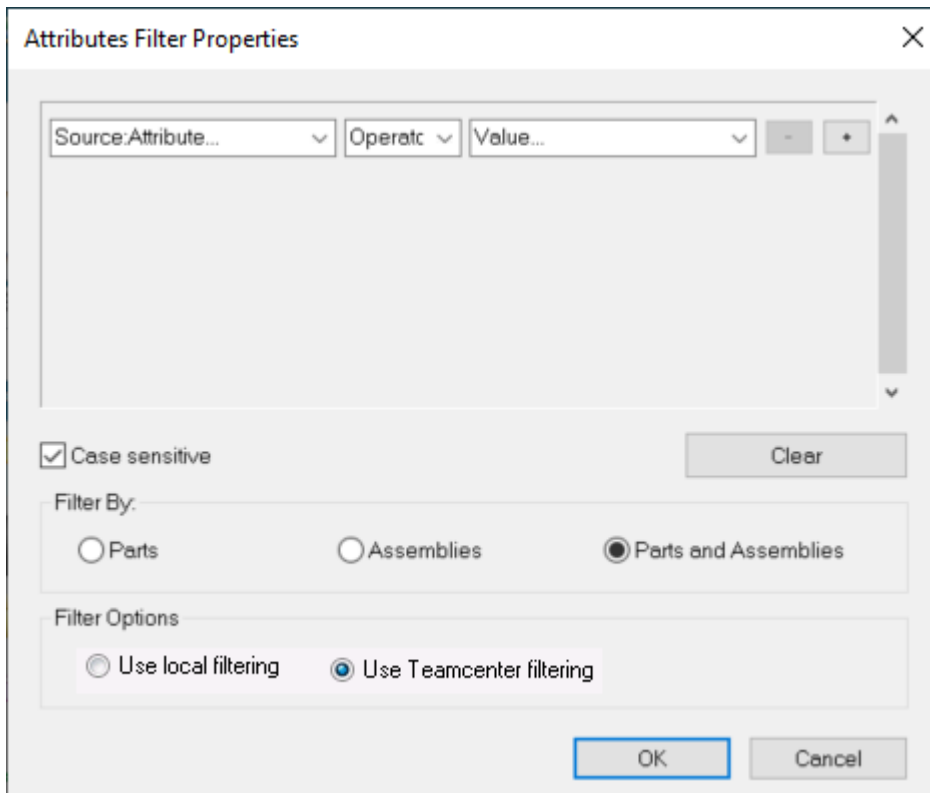
3. In **Filter Sequence**, select an attributes filter.
4. Click **Properties**.

The **Attributes Filter Properties** dialog box displays.

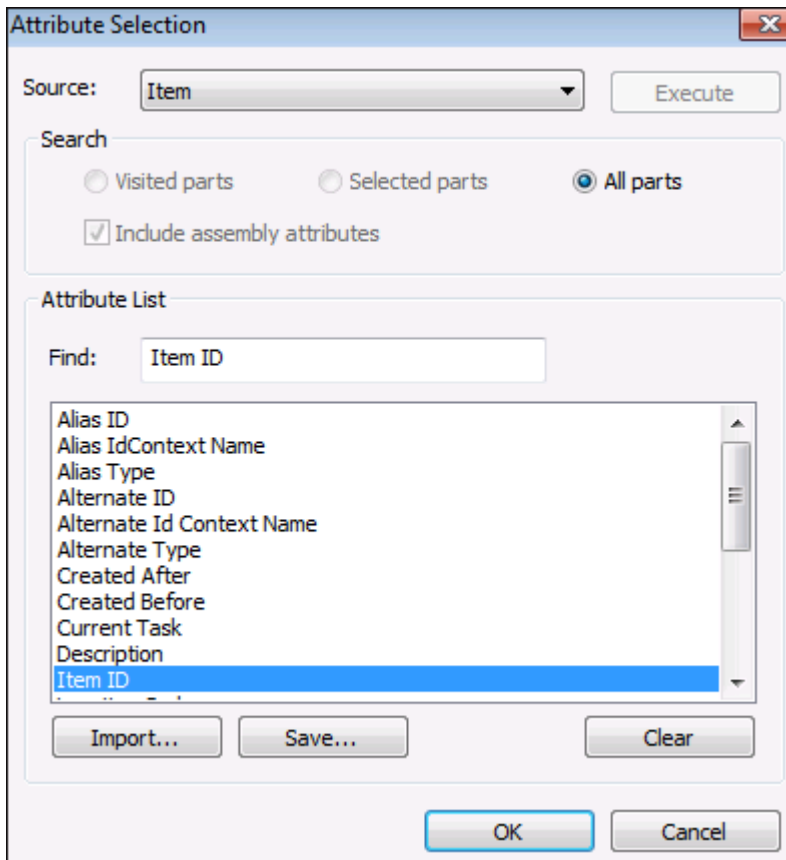
5. In the **Attributes Filter Properties** dialog box, in the **Filter Options** section, choose **Use Teamcenter filtering**, if it is not already selected.

Note:

Cacheless searches are not supported with Workset or 4GD workset data.



6. In the **Attributes Filter Properties** dialog box that appears, select **Source:Attribute**.
7. In the **Attributes Filter Properties** dialog box, in **Source:Attribute**, choose **Select**.
8. In the **Attribute Selection** dialog box that appears, in **Source**, select one of these source options:
 - **Item**
 - **Item Revision**
 - **Occurrence Notes**
 - **Saved Query**



9. Select an attribute from the list, or type the attribute you want in **Find**.
10. Click **OK**.
11. Select at least one part in the Assembly tree.
12. In the **Attributes Filter Properties** dialog box, in **Value**, choose **Select**.
13. In the **Attribute Values** dialog box, select an attribute value in the list, and then click **OK**.

Note:

For best results, use the **Attribute Selection** dialog to select the attribute to filter with.

14. Click **OK**.
15. In the **Filter Manager** dialog box, click **Apply**.

The **Attributes Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

16. (Optional) Click **Revert** to revert the display to its original state. You can switch the display from using the attributes filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Set B-Rep filter properties

If B-Rep (Boundary Representation) data is present in your model, you can filter your model for all parts with or without B-Rep data.

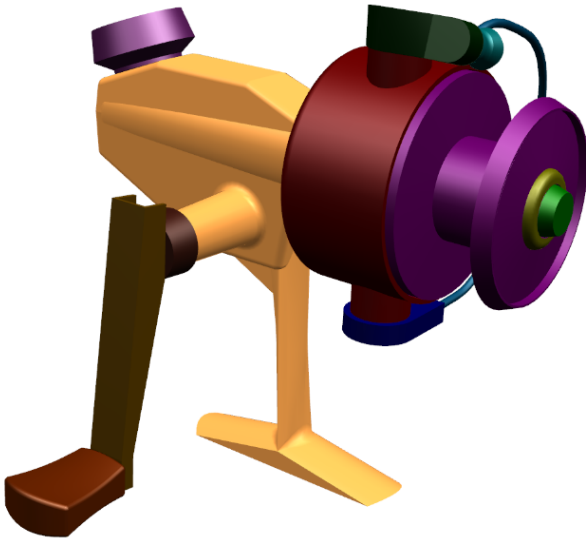
1. (If the Filter Manager is not open) Choose **Actions→Filters**.
2. (If the filter is not on the **Filter Sequence** list) From the **Available Filters** list, select the filter and click **Add**.
3. Select a B-Rep filter in the **Filter Sequence** list and click **Properties**.
4. In the **B-Rep Filter Properties** dialog box, specify to filter for parts with or without B-Rep data.
5. Click **Apply**.

The **B-Rep Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

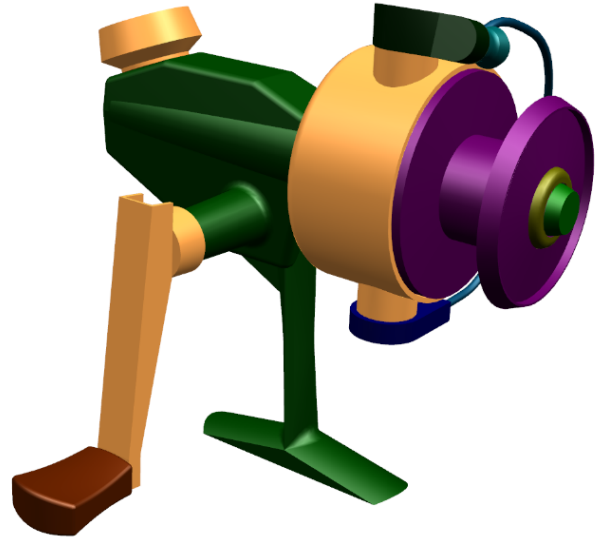
6. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Set Clearance filter properties

The Clearance filter finds all parts that are within a specified distance of a set of target parts.



The Body part is selected as a target part. The distance in the filter is set to 1 mm.



The parts shown in gold are found to be within 1 mm of the target part (Body).

Note:

Because the behavior of the Clearance filter must remain consistent with the other filter types, it uses the **Show Leaf Structure** setting from the **Assembly** view to determine whether to use parts or leaf components to define the filter, not the **Analysis Granularity** in **Clearance Calculator Settings**. If **Show Leaf Structure** is selected, the filter granularity is based on parts (the lowest level geometry available in the product structure). If **Show Leaf Structure** is not selected, the filter granularity is based on leaf components.

1. (If the Filter Manager is not open) Choose **Actions→Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select **Clearance Filter**, and click **Add**.
3. Select **Clearance Filter** in the **Filter Sequence** list, and click **Properties**.

Clearance Filter Properties

Identify Target Parts

☒ Selected ☐ Visible

☐ Include target parts in filter outputs

Filter Parameters

Find all parts within m of targets

☐ Generate List Calculator Settings...

Filter Options

☒ Use local filtering ☐ Use Teamcenter filtering

☐ Apply local Clearance filter (slower)

OK Cancel

4. In the **Clearance Filter Properties** dialog box, in the **Identify Target Parts** section, select an option.

Use this option	To do this
Selected	Filter parts that are within the clearance requirement of all parts that are selected.
Visible	Filter parts that are within the clearance requirement of all parts that are displayed, whether or not they are selected.
Include target parts in filter outputs	Include pairs where both parts are targets in the filter output (but not in the clearance results).

5. In the **Find all parts within** box, type a value for the **clearance requirement**.
6. (Optional) To create a **General Clearance Results** list when you use the Clearance filter, select **Generate List**.

The **General Clearance Results** list is the same list that is created when you create a **clearance list**.

7. (Optional, if you select **Generate List**) To specify the precision and nature of the clearance check, click **Calculator Settings**.

The **Clearance Calculator Settings** dialog box appears.

8. (If you click **Calculator Settings**) Complete the **Calculator Settings**.
9. In the **Filter Options** section, do one of the following:

To	Do this	Speed	Margin of error
Perform a Clearance filter using local tessellated JT data	Choose Use local filtering .	Fastest	$2 \times \text{chordal deviation}$
Perform a cacheless search using True Shape (voxel) data from Teamcenter or Bounding Box data if True Shape data is not available Note: - This option is available only for data loaded from Teamcenter. - Cacheless searches are not supported with Workset or 4GD workset data.	Choose Use Teamcenter filtering .	Medium	$\sqrt{2} \times a$ When using voxel data and "a" is the size of the small cube that forms the true shape as defined by the Teamcenter preference TC_JT_VOXEL_SIZE
	a. Choose Use Teamcenter filtering . b. Select the Apply local Clearance filter (slower) check box. Note: This applies a local Clearance filter (using tessellated data) to the parts found by the cacheless search.	Slowest	

10. Click **OK**.

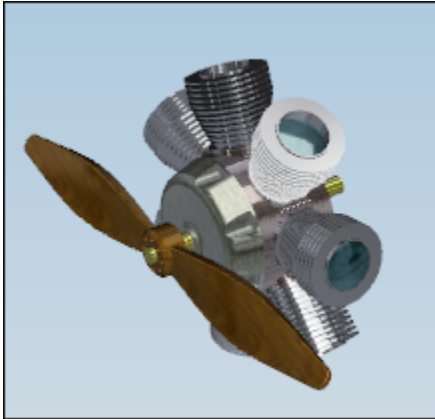
The **Clearance Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

11. In the **Filter Manager** dialog box, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

For more details about searching parts and assemblies, see [Overview of searching assemblies](#).

Set Irregular Zone filter properties

An irregular zone is a *convex hull* or envelope around selected parts. Define the irregular zone to use, and then filter for parts that are inside, outside, or intersecting the zone.



Two selected parts



Irregular zone

1. (If the Filter Manager is not open) Choose **Actions**→**Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the filter and click **Add**.
3. Select an irregular zone filter in the **Filter Sequence** list and click **Properties**.
4. In the Assembly tree or the Viewing window, select parts to define the irregular zone.
5. In the **Irregular Zone Filter Configuration** dialog box, on the **Irregular Zone** tab, click **Go**.
6. In the **Filter Criteria** section, specify how the parts will be tested for intersection with containment in the irregular zone by selecting one of the following options:

- **Part Bounding Boxes**
- **Part Facets**

Note:

Using part facets is more precise, but also slower and more resource-demanding than part bounding boxes. Using part bounding boxes is recommended unless precise results are a requirement.

7. In the **Filter Selected Parts By** section, click an option to define which parts pass the filter.

- **Inside the Irregular Zone**
- **Outside the Irregular Zone**
- **Inside or Intersecting the Irregular Zone**
- **Outside or Intersecting the Irregular Zone**
- **Intersecting the Irregular Zone**

8. (Optional) To control the appearance of the irregular zone geometry, click the **Display** tab, and do any of the following:

To	Do this
Show or hide the surfaces of the zone	Select or clear the Show Surface check box.
Specify the color of the surfaces	From the Show Surfaces color list, select a color.
Show or hide the tessellation lines of the zone	Select or clear the Show Tessellation check box.
Specify the color of the tessellation lines	From the Show Tessellation color list, select a color.
Change the transparency of the zone geometry	Drag the Transparency slider to the left (more transparent) or right (more opaque).

9. Click **Apply**.

The **Irregular Zone Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

10. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Note:

You cannot save Irregular Zone filters.

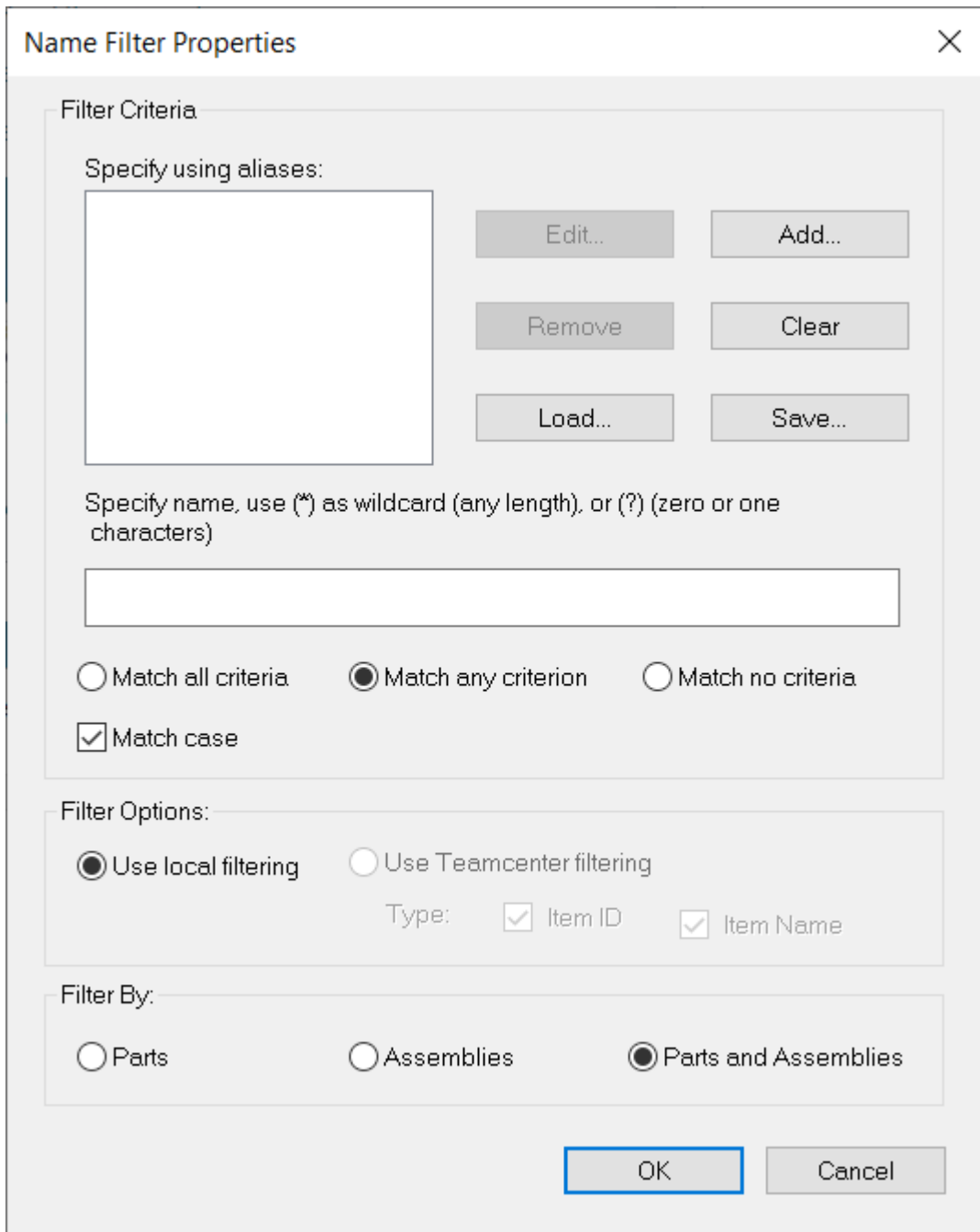
Set Name filter properties

Filter your 3D model by part or assembly name. Only those parts or assemblies matching the defined name filter options will display in the viewing window.

1. (If the Filter Manager is not open) Choose **Actions**→**Filters**.

2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the **Name Filter** and click **Add**.
3. Select a name filter in the **Filter Sequence** list and click **Properties**.

The **Name Filter Properties** dialog box appears.



The **Name Filter Properties** dialog box is shown. It has a title bar with a close button (X). The dialog is divided into several sections:

- Filter Criteria:**
 - Specify using aliases:** A large empty text box for listing aliases. To its right are buttons: **Edit...**, **Add...**, **Remove**, **Clear**, **Load...**, and **Save...**.
 - Specify name, use (*) as wildcard (any length), or (?) (zero or one characters):** A single-line text input field.
 - Match options: ☐ Match all criteria, ☒ Match any criterion, ☐ Match no criteria.
 - ☒ Match case
- Filter Options:**
 - ☒ Use local filtering, ☐ Use Teamcenter filtering
 - Type: ☒ Item ID, ☒ Item Name
- Filter By:**
 - ☐ Parts, ☐ Assemblies, ☒ Parts and Assemblies

At the bottom right are **OK** and **Cancel** buttons. The **OK** button is highlighted with a blue border.

4. (Optional) In the **Specify using aliases** list, select the names of aliases to use with the filter. Do any of the following:

To	Do this
Add new aliases to the Specify using aliases list	Click Add , and then type an alias name and definition. You can use wildcards. You can add multiple definitions, separated by semicolons. The alias file can be stored in a config file in the registry.
Load saved aliases and merge them with the current list	Click Load , and then open a saved alias list (.reg) file.
Save the aliases to (.reg) files for use in future work sessions	Click Save , and then complete the Save dialog box.
Edit an alias	Click Edit and edit the name or definition.
Remove selected aliases from the list	Select an alias name filter and click Remove .
Clear the alias list	Click Clear .

5. (Optional) In the **Specify name, use (*) as wildcard (any length), or (?) (zero or one characters)** text box, type the names to filter.

You can use * or ? as **wildcards** in any position. You can add multiple definitions, separated by semicolons.

6. Select a matching method.

To filter names that	Select
Include all the criteria you have specified	Match All Criteria Note: Only supported by Use local filtering
Include any of the criteria you have specified	Match Any Criterion
Include none of the criteria you have specified	Match No Criteria Note: Only supported by Use local filtering

7. In **Filter Options**, if the data was loaded from Teamcenter **Use Teamcenter filtering** is selected by default.

For name searches, you can also select **Item**, **Item Name**, or both.

Note:

Cacheless searches are not supported with Workset or 4GD workset data.

8. In the **Filter by** section, specify types of names to filter.

To filter by names of	Select
Parts, but not assemblies	Parts Note: Only supported by Use local filtering
Assemblies, but not parts	Assemblies Note: Only supported by Use local filtering
Parts and assemblies	Parts and Assemblies

9. Click **OK**.

The **Name Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

Note:

Using a name filter, when you filter by parts and assemblies, the filter output action is applied only to the topmost matching node of each assembly, even if it has child parts that fit the criteria.

10. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Note:

Important search differences exist between Teamcenter cacheless **Name Filter** and Teamcenter cacheless **Find**. These differences include the following:

- In **Find**, the text string is automatically appended with an asterisk (*) at the beginning and end of the text string. In **Name Filter**, you must manually append the asterisk (*).
- For **Find**, only a single search string is supported. For **Name Filter**, multiple strings are supported.
- Search results vary between these two search conditions. For **Find**, the result only selects the first search result. For **Name Filter**, you can select various output options such as **Select**, **Show**, **Hide**, and so forth.

Set PMI filter properties

Overview of working with PMI filter properties

You can configure the PMI filter to query each model part for specified attributes. A part match occurs when any PMI on a part has an attribute that matches your filter settings.

Tip:

You can combine PMI filters to create a custom **compound filter**.

Set PMI filter properties

1. (If the Filter Manager is not open) Choose **Actions→Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the filter and click **Add**.
3. Select a PMI filter in the **Filter Sequence** list and click **Properties**.
4. In the **PMI Filter Properties** dialog box, from the **Attribute name** list, select an attribute.
5. Click **Add**.

The **Search Criteria List** is populated with the attribute values.

6. (Optional) Add attributes until you are satisfied with the search criteria of the filter.
7. (If you have specified more than one search criteria) Select one of the following options:
 - **Item must match all of these criteria**
 - **Item must match at least one of these criteria**
8. (Optional) Select **Also display PMI that matches this filter**.

Note:

If this option is selected, all of the PMI that matches the search criteria of the filter are displayed. If this option is not selected, only the parts with PMI that matches the search criteria of the filter are displayed.

9. Click **OK**.
10. Click **Apply**.

The **PMI Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

11. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Create an attribute query

1. Under **Attribute Name**, enter a name for the attribute.
2. Click **Add**.

The **Search Criteria List** and the **Available Attributes** list are populated with the name and the values associated with the attribute.

3. Under **Attribute Test** and **Compare to**, enter appropriate values.

Attribute Test	The first Compare To value	The second Compare To value
Exists	Not Available	Not Available
Is Defined	Not Available	Not Available
Equals (Text)	Text. The attribute text must match the specified text (case insensitive).	Not Available
Equals (Numeric)	A numerical value. The attribute value must match the specified value.	Not Available
Greater Than	A numerical value. The attribute value must be greater than the specified value.	Not Available
Less Than	A numerical value. The attribute value must be less than the specified value.	Not Available
Between	A numerical value. The attribute value must be less than or equal to the specified value.	A numerical value. The attribute value must be

Attribute Test	The first Compare To value	The second Compare To value
		greater than or equal to the specified value.

Note:

- When an attribute name is added to the **Search Criteria List**, the default value for **Attribute Test** is **Exists**.
- The values specified for a **Between** query can be in any order. The query will produce the same results, regardless of whether the first value is less than or greater than the second value.
- There is a key distinction in how you use **Exists** and **Is Defined**:
 - **Exists** is true even if the value is *null*.
 - **Is Defined** is true only if the value is not a *null* value.

Modify or delete an attribute query

- (To modify the values associated with an attribute) Select the attribute, and, under **Attribute Test** and **Compare To**, enter the appropriate values.
- (To delete the values associated with an attribute) Select the attribute, and click **Delete**.

Import a PMI attribute list

1. In the **PMI Filter** dialog box, select **Attribute List**.

The **Available Attributes** dialog box appears.

2. Select **Import**.

The **Open** dialog box appears.

3. Browse to the location of the file that contains the attributes that you want to import, select the file, and then click **OK**.

The **Attribute Name** list is populated with the data from your file.

Export a PMI attribute list

1. In the **PMI Filter** dialog box, select **Attribute List**.

The **Available Attributes** dialog box appears.

2. Click **Export**.

The **Save As** dialog box appears.

3. Enter a name for the file.
4. Browse to the appropriate location, and click **Save**.

The list of available attributes is saved as a text file.

Generate the list of available attributes from an active 3D model

1. In the **PMI Filter** dialog box, select **Attribute List**.

The **Available Attributes** dialog box appears.

2. Click **Generate from model**.

The **Generating Attribute List** dialog box appears.

Note:

Generating the list of available attributes may take several minutes to complete. To stop querying the model for attributes, click **Cancel**. If you do cancel this operation, the **Available Attributes** dialog box will be populated with the attributes found before you canceled the query.

Set Visibility filter properties

Note:

The Visibility feature is not supported with **MMV data**.

1. (If the Filter Manager is not open) Choose **Actions→Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the filter and click **Add**.
3. Select a visibility filter in the **Filter Sequence** list and click **Properties**.
4. In the **Visibility Filter Properties** dialog box, select at least one of the following viewpoint options:

To filter for parts that are visible	Do this
In the current Viewing window	Select the Current viewpoint check box.
From multiple viewpoints that you specify	a. Select the Captured viewpoints check box. b. Click Capture Start. A viewpoint is captured, and the number 1 appears next to Captured viewpoints. c. Change the viewpoint by rotating your view or selecting snapshots. As the 3D Viewing window updates, the number of captured viewpoints increases. Tip: - Click Show to display the sequence of captured viewpoints. - Click Clear to remove all captured viewpoints from memory. d. Click Capture End.
From evenly-spaced viewpoints on a sphere surrounding the model	a. Select the Viewpoints on a surrounding sphere check box. b. From the Resolution list, select one of the following: • Low (12 viewpoints) • Medium (42 viewpoints) • High (162 viewpoints) • Super (642 viewpoints)

5. In the **Preferences** section, select the **Use off-screen rendering** check box.

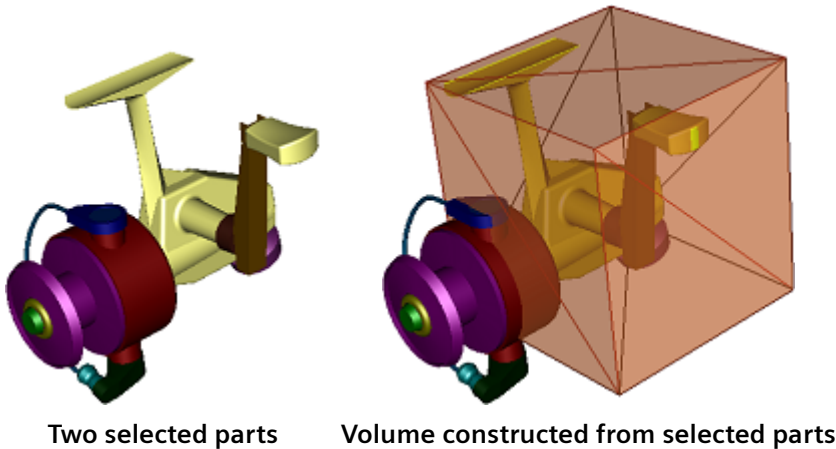
6. Click **Apply**.

The **Visibility Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

7. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Set Volume filter properties

Specify the rectangular-shaped or spherical-shaped volume and whether parts must be completely contained to pass the filter. The rectangular-shaped filter is shown in the example below.



1. (If the Filter Manager is not open) Choose **Actions→Filters**.
2. (If the filter is not in the **Filter Sequence** list) From the **Available Filters** list, select the filter and click **Add**.
3. Select a volume filter in the **Filter Sequence** list click **Properties**.

The **Volume Filter Properties** dialog box appears.

4. (Optional) Create a bounding box around selected parts.
 - a. In the Assembly tree or Viewing window, select the parts.
 - b. In the **Volume Construction** section, click **Rectangular Volume** or **Spherical Volume**.
 - c. In the **Volume Construction** section, click **Go**.
 - d. (Optional) Check the **Enable 3D Manipulator** check box. You can then drag the volume manipulator to resize or reposition the volume.
5. (Optional) In **Display Options**, do one of the following:
 - To hide the volume in the Viewing window, clear the **Display Bounding Volume** check box.

- To display changes in the volume only after you release the mouse button, clear **Interactively Update Volume**.

6. In **Filter Criteria**, specify:

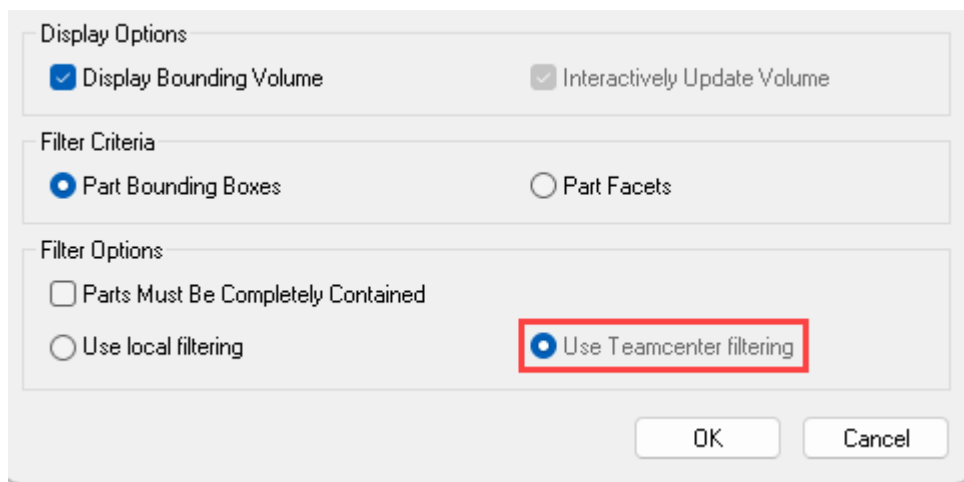
- To calculate using part bounding boxes, select **Part Bounding Boxes**.
- To calculate using part facets, select **Part Facets**.

Note:

Using part facets is more precise, but also slower and more resource-demanding than part bounding boxes. Using part bounding boxes is recommended unless precise results are a requirement.

7. In **Filter Options**, choose a filter option in one of these ways:

- To filter parts that are inside or intersect the volume, clear the **Parts Must Be Completely Contained** check box.
- To filter only parts that are completely inside the volume, select the **Parts Must Be Completely Contained** check box.
- To generate a cacheless search, select the **Use Teamcenter filtering** button.



Note:

- If **Use Teamcenter filtering** is grayed out, the data was not loaded from Teamcenter.
- Cacheless searches are not supported with Workset or 4GD workset data.

8. Click **OK**.

9. In the **Filter Manager** dialog box, click **Apply**.

The **Volume Filter** properties are set. When you apply the filter, parts will be filtered as you have specified.

10. In the **Filter Manager** dialog box that appears, click **Revert** to revert the display to its original state. You can switch display from using the name filter back to the original state of the data by repeatedly clicking **Apply** and then **Revert**.

Create compound filters

Compound filters contain a series of other filters. Apply compound filters as you do single filters.

1. Choose **Actions→Filters**.
2. From the **Available Filters** list, select **Compound Filter** and click **Add**.

The **Compound Filter** is added to the **Filter Sequence** list.

3. **Add** two or more filters to the **Filter Sequence** list.
4. One at a time, drag filters onto the **Compound Filter** until the move arrow points straight left, then release.

The filters are part of the **Compound Filter**.

5. Select the check boxes for filters that you want to apply.
6. Click **Apply**.
7. (Optional) **Save** filters that you want to use later.

Unsaved filters are lost when you close the Filter Manager.

Tip:

Create and save a compound filter when you need to save a series of filters.

Saving filters

If you create filters that you want to use again later, save them before you close the Filter Manager or else you will lose them. Saved filters are added to the Available Filters list in the Filter Manager. Saved filters are stored in the registry of your local system, but you can **export** saved filters to an .xml file that you can share with other users.

Tip:

To save a series of filters, add them to a **compound** filter, and then save it as a single filter.

Note:

You cannot save Irregular Zone filters.

Save filters

1. Choose **Actions→Filters**.
2. **Create** one or more filters.
3. From the **Filter Sequence** list, select the name of a filter to save.
4. Click **Save**.
5. If the **Rename** dialog box appears, type the name for the filter.

Saved filters are added to the **Available Filters** list. They remain on the list until you delete them.

Delete saved filters

You can only delete user-saved filters. You cannot delete the default filters.

1. Choose **Actions→Filters**.
2. From the **Available Filters** list, select a previously saved filter.
3. Click **Delete**.

The filter no longer appears on the **Available Filters** list.

Export saved filters

Saved filters are stored in the registry of your local system. You can share your filters with other users by exporting them to an .xml file. When you export, you export all saved filters. You cannot choose which saved filters to export.

1. Choose **Actions→Filters**.

The **Filter Manager** dialog box appears. If you have saved filters, **Export** is available.

2. (Optional, if you have not saved any filters) **Create** and **save** new filters.
3. Click **Export**.
4. In the **Export Filter** dialog box, browse to the folder where you want to save your filters, name the file, and then click **Save**.

If you save to a file system folder, your saved filters are saved as an .xml file.

If you save to a folder in Teamcenter, your saved filters are saved as an *FndOTcVisFilter* dataset.

Import filters

If other users create, save, and export filters to an .xml file, you can import their filters and use them.

1. Choose **Actions→Filters**.
2. In the **Filter Manager** dialog box, click **Import**.
3. Do one of the following:

To	Do this
Import filters from a file system folder.	In the Import Filter dialog box, select the .xml file to import and click Open .
Import filters from a folder in Teamcenter	In the Import Filter dialog box, select the <i>FndOTcVisFilter</i> dataset to import and click Open .

The imported filters are added to the **Available Filters** list.

Note:

Imported filters are not automatically saved to the registry of your local system. To help you identify unsaved filters, an asterisk is added to the name of each unsaved filter in the **Available Filters** list, for example, ***Attribute Filter Material Aluminum**.

Apply previously saved filters

Filters find parts with specific properties, and then load, display, or select the parts in the Viewing window. You can apply previously saved filters or **create** and apply new filters. Apply a single filter or a series of filters.

1. Choose **Actions→Filters**.

The Filter Manager appears.

2. From the **Available Filters** list, select a **filter** and click **Add**.

The filter is added to the **Filter Sequence** list. This list determines the order in which filters are applied.

3. (Optional) Repeat the process to add more filters.
4. Confirm the **Filter Input Select**, **Filter Output Action**, or **Properties** options for each filter to apply.
5. (Optional) Drag the filters up or down the list to change the filter sequence.

Click the filter name and drag it to a new position.

6. Select the check boxes for filters to apply and clear the check boxes for any filters to ignore.
7. Click **Apply**.

Move filters in the Filter Sequence list

Drag the filters up or down the list.

Click the filter name and drag it to a new position.

Rename filters in the Filter Sequence list

1. Right-click the filter name and choose **Rename**.
2. Type the new name.

Delete filters from the Filter Sequence list

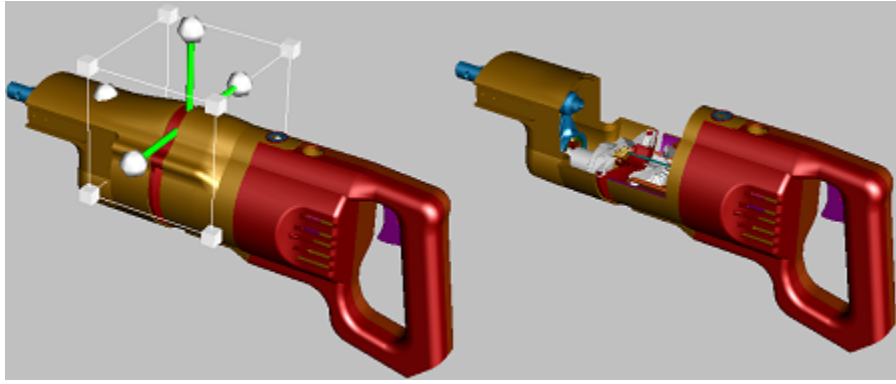
1. Click the filter name.
2. Click **Remove**.

The filter is removed from the **Filter Sequence** list.

Clipping volumes

Clipping volumes overview

Clip areas of a 3D model for a better view of a part or assembly. You can clip areas inside or outside of the clipping volume. Prevent your model from looking hollow by adjusting the clipping properties to include end caps.



Clipping volume limitations

You can use a clipping volume to hide areas of a 3D model. You can prevent selected parts or assemblies from being clipped.

You can have a maximum of only two clipping volumes per view.

The following limitations apply to the simultaneous use of clipping and cross sections:

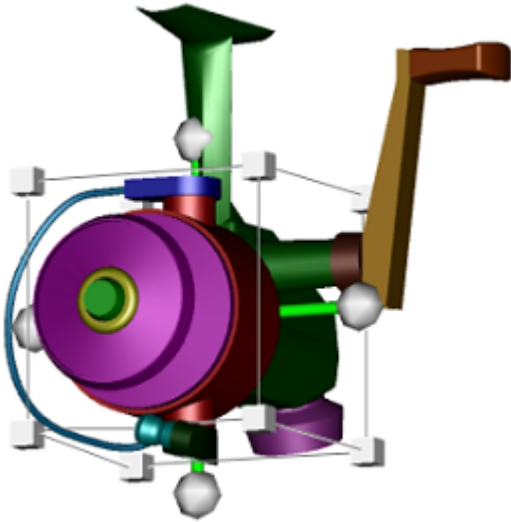
- View Clipping: Clipping outside cannot be enabled when a cross section or cross section 3D clipping plane is present and clipping is enabled.
- Cross Section: Clipping cannot be enabled when a clipping volume is present and clipping is enabled.
- Cross Section 3D: 3D section clipping cannot be enabled when a clipping volume is present and clipping outside of the clipping cube.

Clip areas of models

Clip 3D models for a better view of parts that are hidden by geometry.

1. (Optional) To position the clipping volume around a part, select the part.
2. (Optional) Set **clipping properties**.
3. On the **3D Visibility** toolbar, click **Add** .

A clipping volume manipulator cube appears around the selected part or at the center of the model.



Note:

You can add up to two clipping volumes in the same Viewing window.

4. **Adjust** the clipping volume manipulator to define the volume to be clipped.
5. On the **3D Visibility** toolbar, click **Clip Inside**  or **Clip Outside** .

The model is clipped.

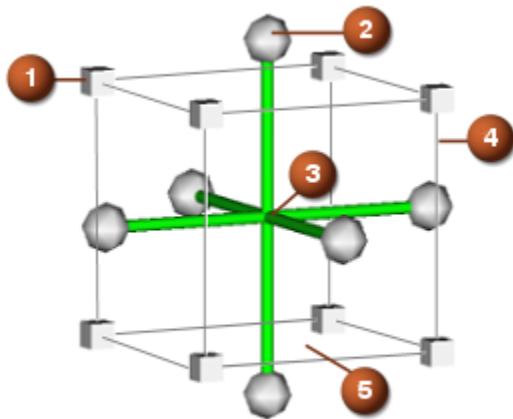
Tip:

To better display the inside of clipped parts, in the View Preferences dialog box, select **Two-sided lighting**.

6. (Optional) To clip two independent sections of your model, repeat with a second clipping volume.

Adjust clipping volume manipulators

Translate, scale, and rotate clipping volume manipulators.



1 Click a square and drag to resize

2 Click a sphere and drag to rotate

3 Rotation point

4 Edge

5 Face

Adjust clipping volumes in one or more of these ways:

To	Do this
Move the manipulator parallel to a face	Drag the face of the manipulator.
Move the manipulator perpendicular to a face	Hold Ctrl as you drag the face.
Constrain the move to one axis of a face	Hold Shift as you drag a face.
Resize the manipulator uniformly along all axes	Drag a corner of the manipulator.
Rotate the manipulator when the green motion jack is hidden	Hold Ctrl and Shift as you rotate the manipulator.
Resize the manipulator along one axis	Hold Ctrl as you drag a corner of the manipulator.
Rotate the manipulator around its X, Y, or Z axis	Drag an edge that is parallel to the axis.
Rotate the manipulator around the X, Y, or Z axis of the green jack	Drag a sphere handle in any direction.
Rotate the manipulator around the center of the green jack, in any direction	Hold Ctrl as you drag a sphere handle in any direction.
Move the green jack (and its center of rotation)	Do one of the following: Right-click a face, edge, or corner of the manipulator, choose Move Jack, and then click anywhere on the manipulator. The green motion jack aligns to the face, edge, or point.

To	Do this
	Hold Shift as you click a sphere handle and drag the green motion jack to the new rotation point.
Align a point on the manipulator with a point on your model	- Right-click the face, edge, or corner of the manipulator that you want to align, and then choose Set Position. - Click a point on your model.
Rotate the manipulator to be parallel to a plane defined by two edges of your model	- Right-click the face, edge, or corner of the manipulator that you want to align, and then choose Set Orientation. - Click two edges.
Move and rotate the manipulator to align it to a point on your model and a plane defined by two edges of your model	- Right-click the face, edge, or corner of the clipping manipulator that you want to align, and then choose Set Composite. - Click a point on your model. - Click two edges.

Hide the manipulator or the green motion jack

You can hide clipping volume manipulators and motion jacks to remove unnecessary objects from the Viewing window.

Tip:

You can rotate the manipulator when the green motion jack is hidden by holding Ctrl and Shift as you rotate.

Hide clipping volume manipulators and motion jacks in the following ways:

To hide or show	Do this
Manipulators and Motion Jacks	On the 3D Visibility toolbar, click Hide Manipulators  .
Motion Jacks only	On the 3D Visibility toolbar, click Hide Jacks  .

Prevent a part or assembly from being clipped

Specify parts or assemblies that cannot be clipped.

- In the **Assembly** view, right-click the part or assembly name, and choose **Clip Off/Clip On**.

An icon  appears to the left of the part name.

When the area inside the clipping volume manipulator is clipped, the part or assembly remains visible.

Remove a clipping volume

1. On the **3D Visibility** toolbar, clear **Clip Inside**  or **Clip Outside** .
2. In the Viewing window, select the clipping volume manipulator.
3. Choose **View**→**Visibility**→**Clipping Volumes**→**Remove**.

Set clipping properties

Adjust the clipping properties to clip geometry inside or outside the manipulator, change the appearance of clipped geometry, and draw end caps.

Note:

- Clipping properties are saved to the registry and are used by default when a new file is opened.
- End capping clipping properties are saved in the session file and are only available in session files that were created using Teamcenter Visualization 5.0 or later.

1. On the **3D Visibility** toolbar, click **Properties** .
2. In the **View Clipping Properties** dialog box, change any of the following clipping properties:

Use this option	To
Properties for clipping: Inside	Clip inside the clipping volume manipulator.
Properties for clipping: Outside	Clip outside the clipping volume manipulator.
Shaded	Display the clipped geometry as shaded.
Transparency	Display shaded clipped inside geometry as transparent. You can specify the color and level of transparency in the 3D viewing Preferences dialog box.
Tessellation Lines	Display the clipped geometry as tessellation lines. Select the line color from the color menu.

Use this option	To
Visible	Display all tessellated lines of clipped geometry.
Invisible	Hide tessellated lines of clipped geometry.
Ghosted	Display tessellated lines of clipped geometry as partially transparent.
Draw End Caps	Cap your model so it does not appear hollow.
Capping Color	Specify the color of end caps.
Set as default	Save the current settings as the default clipping properties

- Click **OK**.

Using virtual reality devices

Interacting with your model

Overview of 3DConnexion devices

The 3DConnexion device translates your sense of touch into dynamic movement of objects with 3D space. It provides optimal manipulation of 3D objects for engineering and animation applications, allowing X, Y, Z pitch, roll, and yaw movement. It provides interactive motion control of 3D graphic objects with up to six degrees of freedom simultaneously; you can zoom, shift, and rotate in one handle.

The 3DConnexion device can be used to move the selected parts when you enable Part Manipulation mode or use the **Part Transformation** dialog box.

You can use a 3DConnexion device to move the current, active cross-section plane. By default, the 3DConnexion device moves the camera, but you can hold the Ctrl key to switch between moving objects or moving the camera. You can also change the default in the **Sensor Properties** dialog box.

You can move the plane in World Coordinate System or in Screen Coordinate System depending on the settings in the **Cross Section Preferences** dialog box. This only works for 3DConnexion devices—not other immersive tracking devices.

You can also use a 3DConnexion device to pan and zoom the image when opening a 2D document. When you load a 2D document and open the **Sensor Properties** dialog box, you can change only the translation sensitivity since the rotation of the device does not have any effect on manipulating the 2D image.

To use the 3DConnexion device, download the driver from the **3D Connexion** website, and follow the instructions in the ReadMe file to install it.

Note:

If the 3DConnexion device buttons do not work with Teamcenter Lifecycle Visualization, update to the latest 3DConnexion driver.

Disconnect the 3DConnexion device

You can disconnect or disable the 3DConnexion device. When you disable the device, you are still connected but the device is not active.

1. Choose **Concept→Sensors→3DConnexion**.
2. In the **Sensor Navigation Properties** dialog box, click **Sensor Properties**.
3. Do either of the following:

To	Do this
Disconnect the 3DConnexion device	On the Connection page of the Properties dialog box, click Disconnect .
Disable the 3DConnexion device	On the General page of the Properties dialog box, clear the Sensor Enabled check box.

Set sensitivity multipliers

When you are using a 3DConnexion device, the distance you move or rotate the device corresponds to translational or rotational movement of the camera, object, or mouse pointer in the work session. You can control the translation and rotational sensitivity for a virtual reality device independently. You can set this sensitivity for each device by increasing or decreasing the value for translation and by increasing or decreasing the rotational sensitivity.

1. Choose **Concept→Sensors→3DConnexion**.
2. In the **Sensor Navigation Properties** dialog box, click **Sensor Properties**.
3. On the 3DConnexion page of the **Properties** dialog box, increase or decrease the values in the **Sensitivity** or **Angular rate** boxes.

Set navigation properties

Use the **Sensor Navigation Properties** dialog box to adjust settings for the 3DConnexion sensor device.

1. Choose **Concept→Sensors→3DConnexion**.

2. In the **Sensor Properties** dialog box, from the **Navigation Style** list, select the **Examine** or **Fly** navigation mode.
3. Click **Orbit Selection** to set the view alignment.

Note:

Orbit Selection works only in the **Examine** mode.

4. In the **Camera Sensitivity Multipliers** section, adjust the values for translation and rotation.
5. If you want to use the 3DConnexion to move parts when in 3D Part Manipulation and Part Transformation modes, select the **Manipulate parts (in Part Transformation and Part Manipulation Mode) and section planes** check box.
6. Choose one of the following:
 - (Default setting) **Manipulate selected object(s). Hold Ctrl key to manipulate camera.**
 - **Manipulate camera. Hold Ctrl key to manipulate selected object(s).**
7. Click **Sensor Properties** then the **3DConnexion** tab to set additional properties.
 - **Sensitivity**
 - **Angular rate**
 - **Translation**
 - **Rotation**
 - **Dominant axis mode**
 - **Reset View**
 - **Device Rezero**

Using stereo viewing

Overview of using stereo viewing

Stereo viewing mode utilizes the stereo rendering capability of your graphics card to display 3D models as stereoscopic images. Shutter glasses filter the stereoscopic output to produce a realistic 3D effect.

Stereo viewing

When you close one eye and look at a scene, you see one view of it; when you close that eye and open the other eye, you see a slightly different view of the scene. A computer can draw both of these images, one after another, so fast that your brain cannot "redraw" the image at the same speed. As a result, the image from the second view is "drawn over" the image of the first view in your brain, resulting in a *stereo view*: you see both images at the same time. This combination of images results in the model appearing to "float" in the air in front of you, acquiring a "real" quality, as a three-dimensional, solid object.

In the program, this effect can be achieved with a number of virtual reality devices. In the image below, the stereo view is achieved through the use of shutter glasses and the model appears to hover outside the computer monitor, between you and the computer monitor.



Shutter glasses

Shutter glasses are a pair of glasses that receive a signal from an emitter box which rests on the top of your workstation screen. The signal from the emitter controls the opening and closing of each of the lenses in the shutter glasses, so that only one lens is open at a time, to create a stereo view of the model for you. With the stereo viewing mode active, the model appears to be located *outside* the computer monitor, hovering between you and the monitor.

Stereo viewing hardware requirements

Stereo viewing mode requires a graphics card that supports stereo rendering, and shutter glasses. The stereo rendering capability must be enabled on your graphics card before attempting to use stereo viewing within the application.

Note:

- Contact your graphics card manufacturer to learn whether your hardware supports stereo viewing.
- On Windows platforms, the graphics card must support quad-buffer stereo mode.
- Enabling stereo rendering on your graphics card can significantly impact the performance of your system. You should enable stereo rendering on your hardware only when you want to use stereo viewing.

Turning on stereo viewing

You can use stereoscopic shutter glasses to obtain a realistic 3D effect when viewing your models, provided your system meets the hardware requirements. You also can use full screen mode to hide the interface and display your model using the entire screen of your monitor.

Turn on stereo viewing

1. On the **3D Display Devices** toolbar, click **Stereo Viewing Mode** .

Your workstation screen goes momentarily black as the application switches to the stereo viewing mode, and then a **Confirmation** dialog box appears.

2. (If the model is displayed correctly) Click **Yes** to close the dialog box.
3. (If the model does not display correctly) Click **No** to exit stereo viewing mode, and then ensure that you meet the hardware requirements for stereo viewing before attempting to turn on stereo viewing mode again.

Tip:

When you are sure that your system supports the stereo viewing mode without problems, you can disable the **Confirmation** dialog box within the stereo viewing preferences.

4. When you no longer want to view your model in stereo viewing mode, on the **3D Display Devices** toolbar, click **Stereo Viewing Mode**  to return to the normal 3D Viewing window.

Turn on full screen mode

1. On the **3D Display Devices** toolbar, click **Full Screen Mode** .

The Viewing window enlarges to fill your entire monitor.

Tip:

Use the following keyboard shortcuts in conjunction with the mouse to manipulate your model in the Viewing window:

Use this key	To do this
Ctrl	Pan the model
Shift	Zoom the model in or out

- When you no longer want to view your model in full screen mode, press Esc on the keyboard.

-or-

In the Viewing window, right-click and choose **Display Devices**→**Full Screen Mode**.

Disable the stereo viewing Confirmation dialog box

- On the **3D Display Devices** toolbar, click **Preferences** .
- On the **General** tab of the **Display Device Preferences** dialog box, clear the **Show confirmation dialog when switching into stereo mode** check box.

Adjust the stereo viewing effect

You can make the 3D effect more pronounced by adjusting the offset between the stereo images.

Tip:

Turn on **Dynamic Eye Separation** to dynamically change the eye separation. If you set up a proper eye separation and then navigate into a small area of the model, the eye separation may be too large to comfortably view the parts, which may result in eye fatigue. This option adjusts the eye separation value dynamically based on the view distance, giving you a more comfortable stereo viewing experience, although the geometric perception may not be correct. You can turn on this option only when in the stereo viewing mode.

- On the **3D Display Devices** toolbar, click **Preferences** .
- In the **Display Device Preferences** dialog box, in the **Eye Separation** section of the General page, select the **Relative** or **Absolute** check box.
- Do one of the following:

- Move the **Eye Separation** slider to increase or decrease the offset of the stereo view.
- Type an offset amount, and then click **Apply**.

The larger the offset, the more dramatic the stereo view appears and the closer the model appears to you. The smaller the offset, the flatter the model appears, until it becomes nearly two-dimensional.

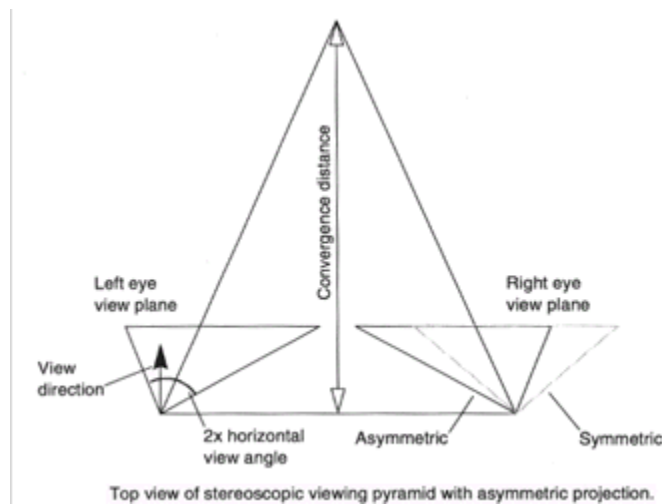
4. (Optional) Turn on **Dynamic Eye Separation**.
5. On the Projection page, move the **Converge Distance** slider to set the convergence distance of the viewpoint.
6. Move the **Field of View** slider to set the field of view.

Overview of stereo viewing convergence distance

Convergence distance measures the distance from the view angle to the point where the left eye view plane and the right eye view plane meet. You can set a value for the convergence distance of a viewpoint.

Note:

For this value to have any effect on your scene, you need to turn on asymmetric viewing.



Asymmetric viewing is useful in some stereo viewing configurations. By changing the convergence distance, geometries can be made to appear either in front of or behind the display device. A geometry from a viewpoint that is less than the convergence distance appears in front of the screen. A geometry that is farther away from the viewpoint appears behind the screen.

You can achieve a dramatic stereo effect when part of your scene appears to be in front of the screen and part appears to be behind the screen. Experiment with setting the convergence distance to the distance from the viewpoint to the midpoint in your scene for the results you want.

Viewing JT files containing watermarks

Overview of JT files containing watermarks

You can view JT files containing watermarks authored using Teamcenter Convert 8.3, or later. Watermarks consist of text strings or images that provide a degree of proprietary ownership for assemblies and parts that you share with others.

Note:

- Watermarks are not markup data. As a result, they do not generate product structure because the watermark is an inherent part of the data.
- Watermarks can be exported using JT file version 9.5 or higher.
- If you enable **page extents**, watermarks automatically scale to fit the page.

Export JT watermark files

1. Open a JT file containing a watermark.
2. Choose **File→Export**.
3. In the **Export** dialog box, from the **File format** list, select **DirectModel (*.jt)**.
4. Click **Options**.
5. In the **JT Export Options** dialog box, in the **File Format** section, select **9.5** (or higher) from the **JT Version** list.
6. In the **Export** dialog box, verify the **File** location is correct, and then click **OK**.

The JT watermark file is exported to the location specified.

Watermarks and page extents

If you enable page extents, watermarks automatically scale to fit the page.

1. Open a JT file containing a watermark.
2. Choose **View→Page Extents**.

3. In the **Page Extents** dialog box, click the **Enable Page Extents** check box.
4. In the **Select the active page extents** section, select the page extents to use or click **Add** to add a new page size.
5. (Optional) Click the **View Visual Guide** check box.
6. Click **OK**.

2. Searching 3D models

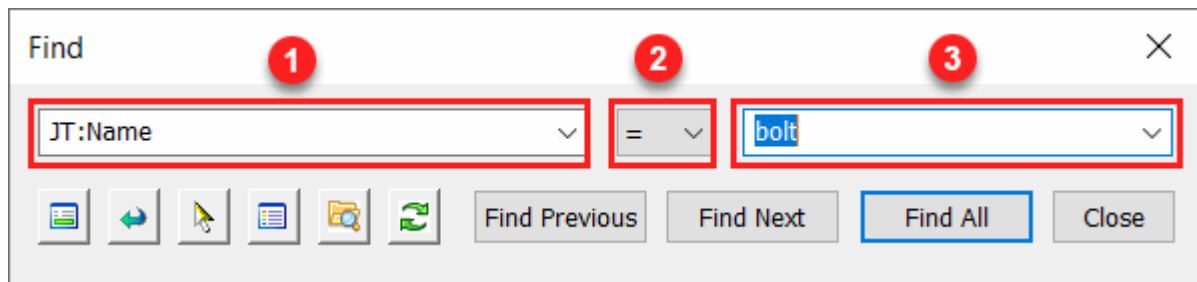
Find parts based on a property value

When Visual Reporting is installed, you can use the advanced **Find** command to find parts and assemblies with property values that match a search pattern and then apply an action to them.

Example:

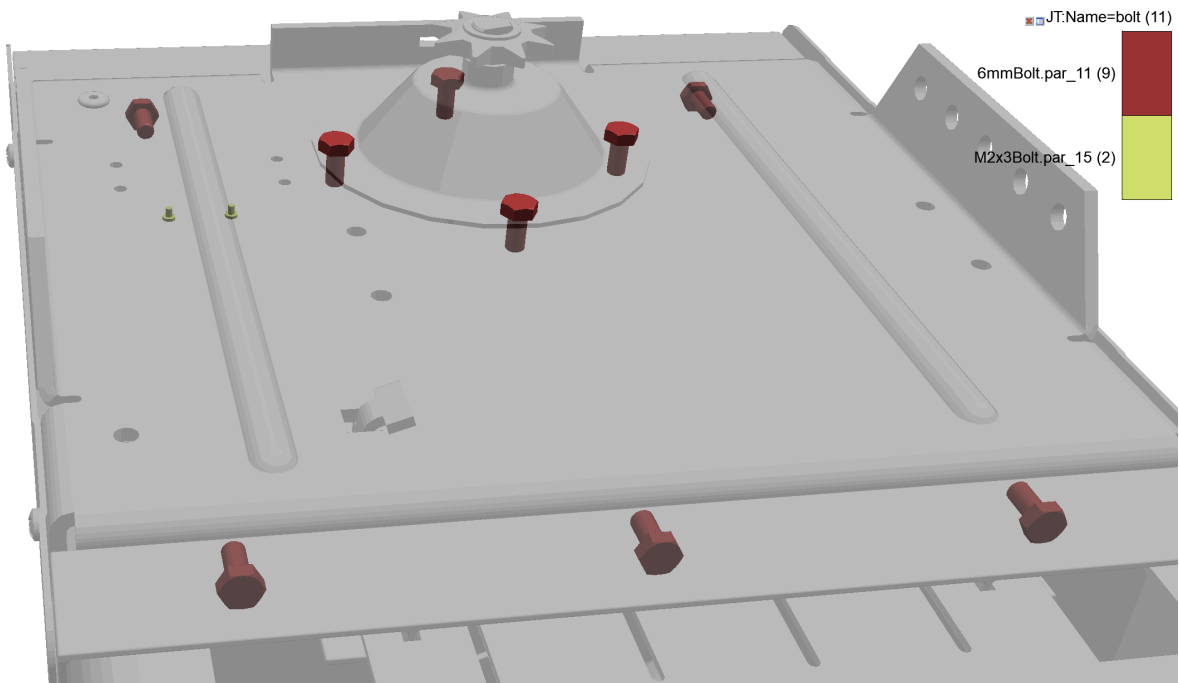
In this example, a designer wants to find all bolts in a model.

1. In an empty area of the Viewing window, right-click and choose **Find**, or choose **Home** tab→**Visual Report** group→**Find** .
2. In the **Find** dialog box, specify **JT:Name = bolt**.



1) Property 2) Condition 3) Value

3. Click **Find All**.



All parts with the JT attribute **Name** where the value includes "bolt" are found and an action can be applied, such as selecting all matching parts. In this case, occurrences of parts **6mmBolt.par_11** and **M2x3Bolt.par_15** match the search pattern, and each occurrence group is colored with a group color. Parts that do not match the pattern are colored transparent gray.

Tip:

Create a snapshot or a group to save results of advanced **Find** actions.

Procedure

1. Start the command in one of these ways:

For this initial property availability	Start the command this way
Properties and values from the first 100 nodes in the product structure	In the Viewing window, right-click an empty area and choose Find , or choose Home tab→ Visual Report group→ Find  .
Properties and values from a particular node or part, with the ability to peruse and copy properties and values from the Properties dialog box for the node or part	Right-click a node or part and choose Properties , and then in the Properties dialog box, click Find . While both dialog boxes are open, you can click an attribute name in the Properties dialog box and the property and its value are copied to the Find dialog box.

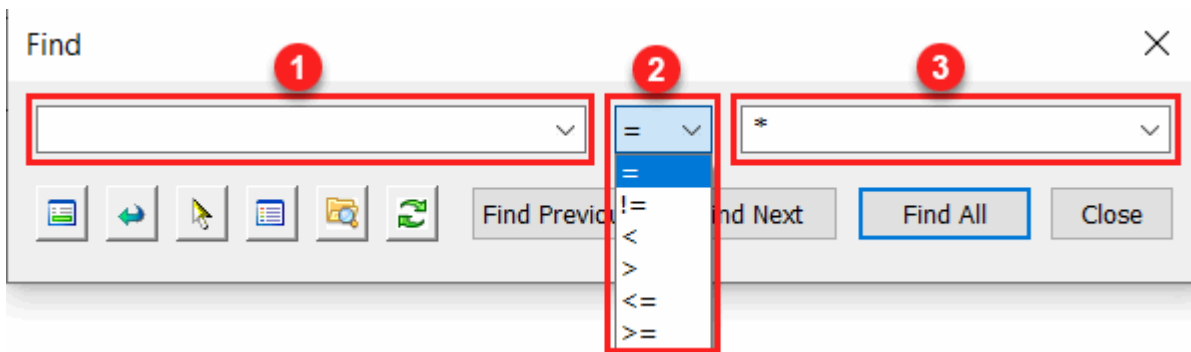
For this initial property availability	Start the command this way
	Mouse navigation modes are not available while the Properties dialog box is open. When the Find dialog box is opened from the Properties dialog box, closing the Properties dialog box also ends the Find command.

2. As needed, specify what you want to do with the Find operation. Click **Preferences**  to open the **Find Preferences** options dialog box.

To do this	Select this Option
Limit parts to include in the search.	Input Choose an option from the list.
Choose the action to apply to the search results.	Action • Show only (Loads matching parts) • Show additive (Loads matching parts) • Hide • Select only • Select additive • Deselect • Change appearance (Loads matching parts) You can click More Options to specify how the appearance of matched and unmatched results should change.
Include assembly nodes in the results, in addition to parts.	Include assembly nodes
Zoom the view window to the results of the search.	Zoom to matching results

To do this	Select this Option
Match only property values that have only the specified pattern characters.	Match whole word only
Match only property values where the character case matches the specified pattern character case.	Match case
Create a visual report of the results.	Click More Options to open the Visual Report Preferences dialog box and specify options for creating a visual report of the results.

3. In the **Find** dialog box, specify parameters for the property pattern you want to find.



For this pattern parameter	Do this
1) Property	Enter or select the property for the search pattern. Note: To refresh the list from properties of the currently selected nodes or parts, click Refresh Attributes List . If no nodes or parts were selected, a message appears prompting you to select a node or part.
2) Condition	Select the condition for comparing the property value.

For this pattern parameter	Do this
	Note: For text values, only the = (contains) and != (doesn't contain) conditions are valid.
3) Value	Enter or select the value to search. The value can be either numbers or text. Note: When the value box (3) contains an asterisk (*), all parts and assemblies that have the selected property match the search pattern.

4. Click a **Find** command to find the matching occurrences and apply the specified actions.

- **Find Previous** — returns focus to a previous matching occurrence
- **Find Next** — sets focus to the next matching occurrence
- **Find All** — applies actions to all matching occurrences

Additional options

To do this	Click this command
Clear the applied action(s).	Clear  If no visual report was applied to the assembly, a message appears prompting you to reset the appearance to the default materials. If a visual report was applied to the assembly, the appearance is reset to the state before the report was applied.
Select all matching parts found by the search.	Select Matching Parts 
Display an interactive legend for the search results.	Results Legend 

To do this	Click this command
	For information about manipulating display of results in the Results Legend dialog box and in the Viewing window, see Examine results of the visual report .
Cycle through reports for properties or values (available in Windows OS only).	Explore Mode  and then: - Click either the property box or the value box to set focus on that parameter. - Press the up or down arrow key to choose the next item in the parameter list. Note: If focus is on the property box, when you press an arrow key to choose a new property, the value box automatically is set to asterisk (*) so that all values for the property are matched.
Refresh the properties and values lists.	Refresh Attributes List  Refreshes the lists from properties of the currently selected nodes or parts. If no nodes or parts were selected, a message appears prompting you to select a node or part.

Search the assembly tree of a local .jt file

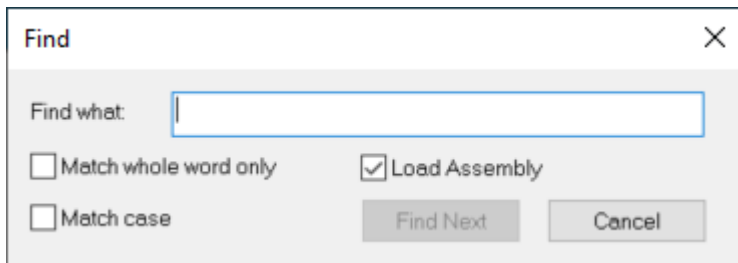
This topic describes the **Find** dialog, which allows you to search part and assembly names in the assembly tree of a .jt file that was opened from the local file system. Different versions of this dialog are available, depending on the service level. If you are using Mockup, you can also use the **3D advanced search functionality**, which allows you to find parts and assemblies using names or metadata and display the matching results in the Viewing window using a visual report.

Base, Standard and Professional

The number of search options in the **Find** dialog box varies for the different viewer service levels. The following describes how to search the assembly tree if you are using Base, Standard, or Professional.

- Open the **Find** dialog box in one of these ways:
 - Right-click in the assembly tree and choose **Find**.
 - On the **Tree Control** toolbar, click **Find** .

The **Find** dialog box appears.



2. (Optional) Check the **Match whole word only**, **Match case**, or **Load Assembly** check boxes.
3. In the **Find what** section, enter a text string to find.
4. Click **Find Next**.

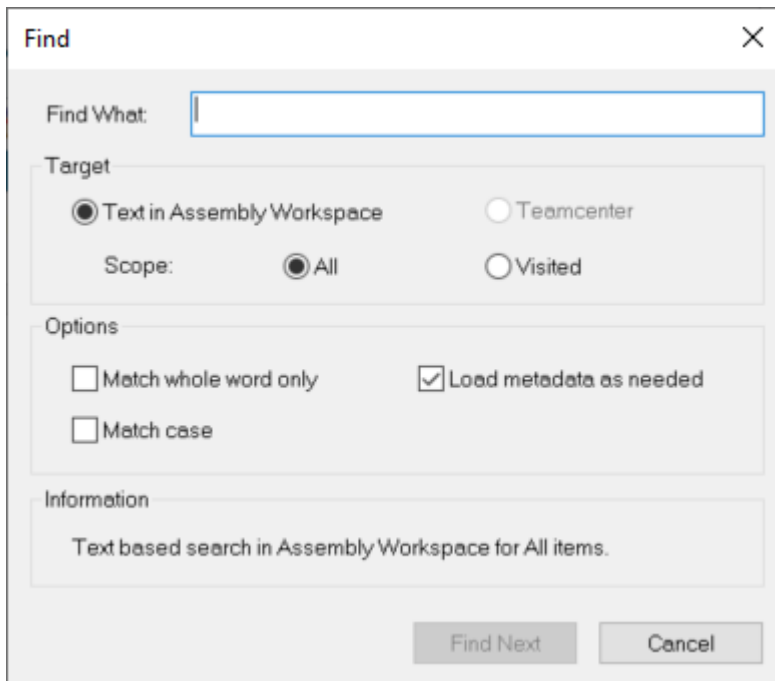
The first entry that contains the search text string is selected. It is highlighted in the assembly tree. Click **Find Next** to find other entries.

Mockup

There are more search options at the Mockup viewer service level. The following describes how to search the assembly tree if you are using Mockup.

1. Open the **Find** dialog box in one of these ways:
 - Right-click in the assembly tree and choose **Find**.
 - On the **Tree Control** toolbar, click **Find** .

The **Find** dialog box appears.



When you search the assembly structure of a .jt file that was opened from the local file system, the following options are selected as the default settings: **Text in Assembly Workspace**, **All**, and **Load metadata as needed**.

2. In the **Target** section, select **All** to search all parts and assemblies, or select **Visited** to search all visible parts.
3. (Optional) Check the **Match whole word only**, **Match case**, or **Load metadata as needed** check boxes.
4. In the **Find what** section, enter a text string to find.
5. Click **Find Next**.

The first entry that contains the search text string is selected. It is highlighted in the assembly tree. Click **Find Next** to find other entries.

Search an assembly saved in Teamcenter

Use **Find** to locate a part or assembly in a model that is saved in Teamcenter and opened in the Lifecycle Viewer or in the integrated standalone application viewer. You can search for **Item ID** only, **Item Name** only, or both of these objects.

1. From My Teamcenter, send an assembly to the Lifecycle Viewer or to the integrated standalone application viewer.
2. Open **Find** in one of these ways:

- Right-click on a specific component and select **Find**.
- Right-click anywhere in the Assembly tree and select **Find**.
- Choose **Tools**→**Assembly**, and when the **Assembly** menu appears, choose **Assembly**→**Find**.

The screenshot shows the 'Find' dialog box with the following settings:

- Find What:** (Empty text field)
- Target:**
 - ☐ Text in Assembly Workspace
 - ☒ Teamcenter
- Type:**
 - ☒ Item ID
 - ☒ Item Name
- Options:**
 - ☐ Match whole word only
 - ☒ Load metadata as needed
 - ☐ Match case
- Information:**

Teamcenter Search for Item ID or Item Name.
- Buttons:** Find Next, Cancel

The **Teamcenter** option is only available when the **Find** dialog is accessed by right-clicking on a component that was loaded from Teamcenter.

3. (Optional) In the **Type** section, check the **Item ID** and the **Item Name** check box, or only check one of them.

When both check boxes are checked, the processing searches for an item ID and an item name.

If you know the general nature of the search, checking only the **Item ID** or **Item Name** check boxes may improve the search performance.

Example:

Search string conditions are very flexible. When you select **Item ID** and **Item Name** and then load subassembly 33000048/A:1-drag-assembly the following search inputs are valid:

- 33000048

- drag-assembly
- drag
- assembly
- assem
- 33000
- 0048
- 3000

The text string before /A is the **Item ID** and the text string after /A is the **Item Name**. The symbol /A indicates that this is Revision A of this object.

The wild card character (*) is automatically added to the beginning and the end of the string. If you do not want to add the wild card, check the **Match whole word only** check box in the **Find** dialog box.

4. Click **Find Next**.

The first entry that contains the text string is selected. Click **Find Next** to find other entries.

Create filters to load, display, or select parts

1. Choose **Actions→Filters**.
2. From the **Available Filters** list in the Filter Manager, select a **filter type** and click **Add**.

The filter is added to the **Filter Sequence** list. This list determines the order in which filters are applied.

3. From the **Filter Sequence** list, select the filter.
4. From the **Filter Input Select** list, specify which parts to filter:

To filter	Select
All parts in the assembly, whether or not they are loaded	All Parts
Parts that are loaded, whether or not they are displayed	Loaded Parts
Parts that are displayed	Visible Parts

To filter	Select
Parts that are selected	Selected Parts
Parts that pass the previous filter in the Filter Sequence list	Previous Output

- From the **Filter Output Action** list, specify what happens to the parts that match the filter criteria.

To	Select
Make no changes to the part state as a result of the filter	None
Display only the parts that pass the filter and hide all parts that do not pass the filter	Show Only
Display the parts that pass the filter, in addition to parts that are currently displayed	Show Additive
Hide the parts that pass the filter	Hide
Select the parts that pass the filter and deselect all parts that do not pass the filter	Select Only
Select the parts that pass the filter, in addition to parts that are currently selected	Select Additive
Deselect the parts that pass the filter	Deselect

- Click **Properties** and **set the filter properties**.
- (Optional) To rename the filter, right-click the filter and choose **Rename**.
- Select the check boxes only for filters that you want to apply.
- Click **Apply**.

The filter output action is applied.

Note:

If you create a **Name** or **Attributes** filter, and you filter by parts and assemblies, the filter output action is applied only to the topmost matching node of each assembly, even if it has child parts that fit the criteria.

- (Optional) **Save** filters that you want to use later.

Note:

Unsaved filters are lost when you close the Filter Manager.

11. (Optional) Export saved filters to a file.

Create part reports

Use HTML part reports to list specific attributes for parts of your model.

1. Choose **Actions**→**Part Report**.
2. In the **Part Report-Part Set Selection** dialog box, select from the following:

Choose this option	To do this
Visible	Create a report that includes information for all visible parts.
Selected	Create a report that includes information for selected parts.
With text markups	Create a report that includes information for only those parts that have anchored text markups. Select Include Identifier to use the text in the anchored text markup as the part identification.
Parts (Always On)	Perform analysis on all parts in the model. Note: You cannot disable part-based analysis. However, you can specify for leaf components and end items to override parts.
Leaf Component	Perform analysis on leaf components in the model.
End Item	Perform analysis on end items in the model.

3. Click **Create Report**.
4. In the **Report Definition Wizard** dialog box, follow the remaining steps.

Display PMI types using filters

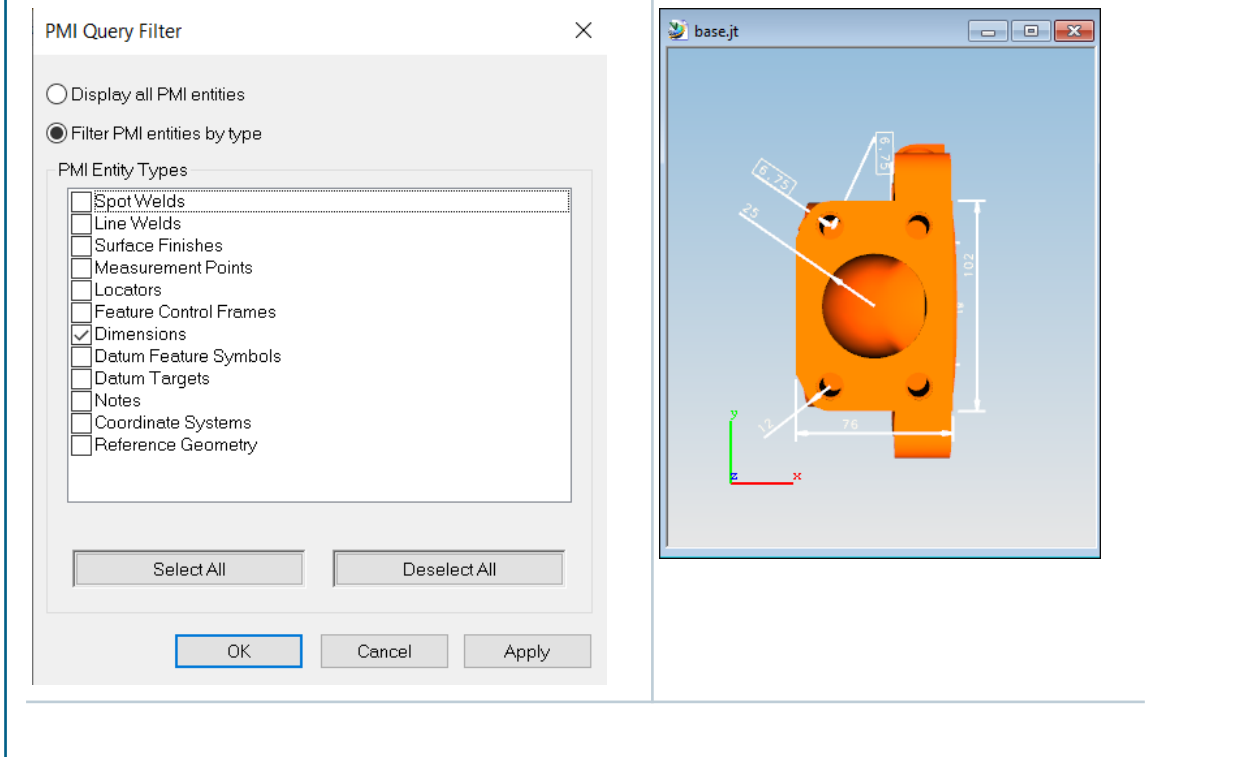
Use PMI filters to specify the PMI entities you want to display in the Viewing window.

1. On the **PMI** toolbar, click **Filtering** .

2. Choose **Filter PMI Entities by Type** from the **PMI Query Filter** dialog box.
3. To display or hide each PMI type, select or clear **PMI Entity Types**.
4. Click **OK**.

Example:

In this example, you only want to display **Dimensions** PMI.



Creating visual reports

Overview of visual report expressions and rules

Use expressions to set visual report conditions that provide the results you want. Visual reports use standard regular expressions to define Boolean syntax. An expression consists of one or more sets of terms and each term is a property value comparison. For example, if you need a visual report that displays parts that weigh less than 2.5 kg and that were designed by Joe, the expression is `Name=Joe AND Weight<2.5`.

For visual reports, expression rules and definitions include the following:

- Special characters and operators

- Details about condition expressions
- Pattern strings
- Date formatting
- Meta-variables

Special characters and operators

You use regular expressions to specify the pattern used in a search. The following table defines common special characters and operators that you can use in your regular expression syntax.

Use this	To do this
Asterisk (*) special character	Match zero or more characters.
Question mark (?) special character	Match zero or one character.
Backslash (\) special character	Interpret the next character literally. Example: The pattern <code>foo*</code> matches <code>foo*</code>, but does not match <code>food</code>.
Double quotes (") special character	Delimit strings. Special characters inside the quotation marks are treated as special characters. Example: The pattern <code>"John Doe"</code> matches <code>John Doe</code>, <code>John Doe\</code>, and <code>John Doe*</code>.
Single quote (') special character	Delimit strings. Special characters inside the quotation marks are treated as literal characters. Example: The pattern <code>'John'</code> matches <code>John</code> but not <code>John</code> or <code>John Doe</code>.
Comparison operator	Compare the left side property to the right side property.

Use this	To do this
	Note: For more details, see details about condition expressions.
Logical operator	Combine multiple terms into a complete expression. Note: You can type logical operators as text or symbols. If you type them as text, make sure you use capital letters. Logical operators include AND & OR NOT !. Example: Name=Joe AND Color=red Name=Joe & Color=red Color=green OR Color=red Color=green Color=red Name=Bill AND NOT (Weight<2.5 OR Color=green)
Parentheses [()] special character	Change the order of precedence or use them to group logical terms together.
Dollar sign (\$) operator	The meta-variable operator.

Details about condition expressions

The basic syntax form for conditional expression interpretation is:

<key> <op> <value>

where <key> (interpreted as the name of the property) and <value> (interpreted as the target value of the property) are character strings and <op> is one of the following comparison operators:

<	Less than
>	Greater than
=	Equal to or matches the value
<=	Less than or equal to
>=	Greater than or equal to

!= Not equal to or does not match

Three basic value types are supported:

- Date
- Number (integer or floating point)
- String

Note:

- Simple string values are only compared with = (equal to) or != (not equal to) operators.
- If you compare simple string values with less than or greater than operators, the expression evaluates to `false`.
- For date type values, you can use + or – and an integer to add or subtract days, for example `[Today]-3`.

Pattern strings

A pattern string is one where the expression contains special characters * or ? that are not escaped in the expression. Use pattern strings to identify partial string matches. You can type the special characters on either the left or right side of the operator.

Example:

Color = bl*	Matches the color blue and black, but not navy-blue
* = John*	Matches Owner=John and Author=Johnson, Bill
Name = ?il*	Matches Name=Bill and Name=William, but not Name=Philip

Date formatting

Visual reports support several date formats. These formats are not influenced by the locale in which the application is running. Visual reports support three date field separation characters: slash (/), dash (–), and period (.).

Example:

For most U.S. English locales `mm/dd/yyyy`

For many Asian-Pacific locales and ISO 8601 `yyyy-mm-dd`

For many European locales `dd.mm.yyyy`

If required, you can also include a time specification. The time is separated from the date with either a colon (:), dash (–), or space (). But, the time syntax is always `hh:mm:ss`.

Meta-variables

When you want to test the value of one property against the value of another property you can use meta-variable syntax. Use the dollar sign (\$) to indicate the meta-variable.

Example:

```
WEIGHT < $TARGET_WEIGHT
```

This means the visual report evaluates each part for the `WEIGHT` property and the value of the `TARGET_WEIGHT` property and then compares these values. The report then displays parts that weigh less than the target weight.

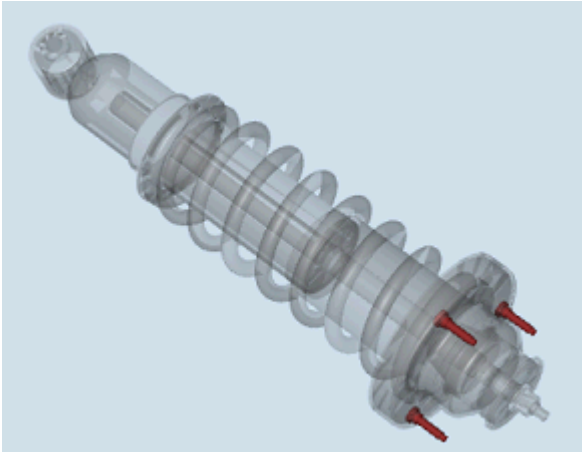
Create a quick visual report

Use **Quick Report** to generate a visual report of components with attribute keys and values that match a specified text expression.

Example:

In this example, a designer wants a visual report of *shock_absorber.jt* parts where the part name starts with `st*`.

1. On the **3D Visual Report** toolbar, click **Quick Report** .
2. In the **Quick Report** dialog box, type **"part name" = st*** and click **Apply**.



All parts where the part name starts with **st** are displayed in the default matched parts material (dark red). All other parts are displayed in the default unmatched material (transparent gray).

Tip:

You can also use the **Find** command to easily create a similar report.

1. On the **3D Visual Report** toolbar, click **Quick Report** .

The **Quick Report** dialog box opens.

2. Type your visual report expression in the text section of the **Quick Report** dialog box.
3. Click **Apply** to generate the visual report and display the results in the Viewing window.
4. (Optional) Click **Preference** to open the **default preferences** options.

In the **Preference** dialog box, you can specify the default colors for matched and unmatched parts. You can also change how the application searches for strings. Select either case insensitive (the default) or case sensitive.

5. (Optional) Click **Save As** to open the **Save Visual Report File** dialog box and save the quick report as a .vpx file.

The quick report name also appears in the **Reports** section of the **Apply Report** dialog box. This dialog box supports **loading and managing** saved visual reports.

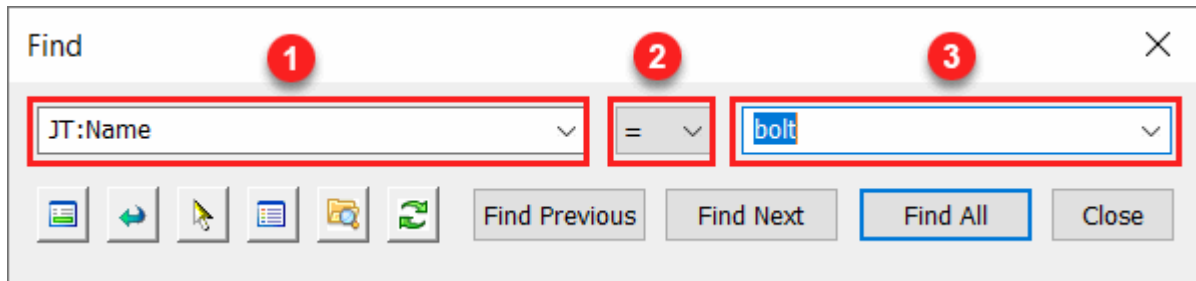
Find parts based on a property value

When Visual Reporting is installed, you can use the advanced **Find** command to find parts and assemblies with property values that match a search pattern and then apply an action to them.

Example:

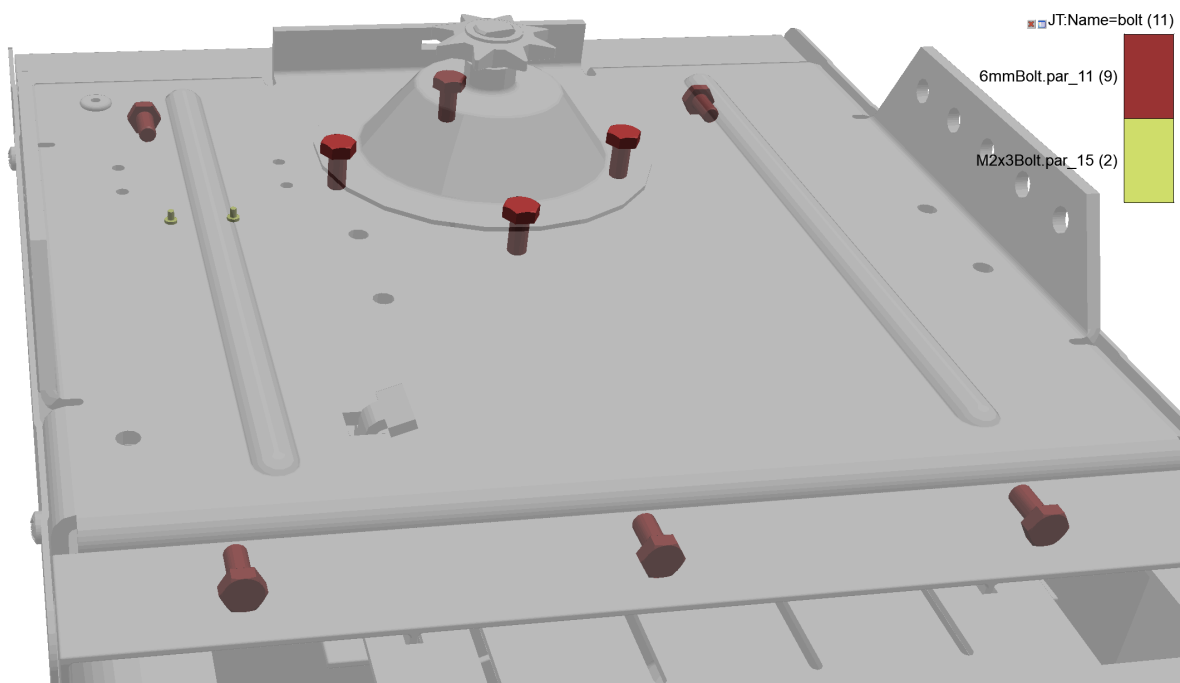
In this example, a designer wants to find all bolts in a model.

1. In an empty area of the Viewing window, right-click and choose **Find**, or choose **Home** tab→**Visual Report** group→**Find** .
2. In the **Find** dialog box, specify **JT:Name = bolt**.



1) Property 2) Condition 3) Value

3. Click **Find All**.



All parts with the JT attribute **Name** where the value includes "bolt" are found and an action can be applied, such as selecting all matching parts. In this case, occurrences of parts **6mmBolt.par_11** and **M2x3Bolt.par_15** match the search pattern, and each occurrence

group is colored with a group color. Parts that do not match the pattern are colored transparent gray.

Tip:

Create a snapshot or a group to save results of advanced **Find** actions.

Procedure

1. Start the command in one of these ways:

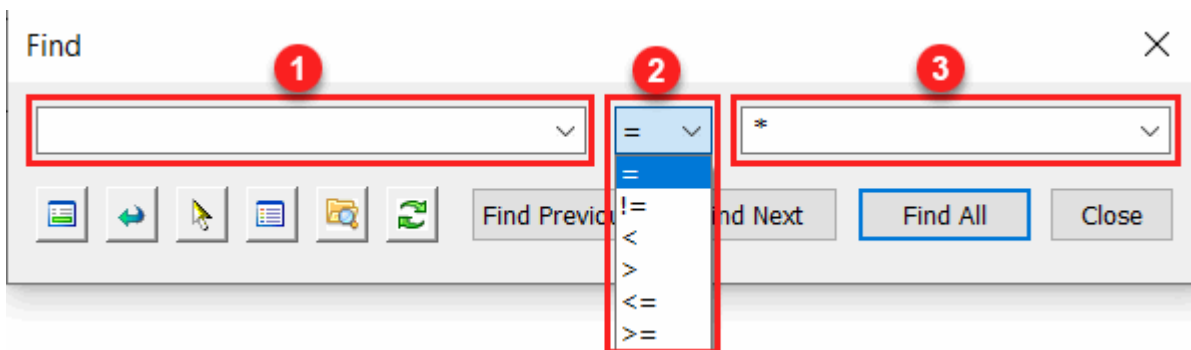
For this initial property availability	Start the command this way
Properties and values from the first 100 nodes in the product structure	In the Viewing window, right-click an empty area and choose Find , or choose Home tab→ Visual Report group→ Find  .
Properties and values from a particular node or part, with the ability to peruse and copy properties and values from the Properties dialog box for the node or part	Right-click a node or part and choose Properties, and then in the Properties dialog box, click Find. While both dialog boxes are open, you can click an attribute name in the Properties dialog box and the property and its value are copied to the Find dialog box. Mouse navigation modes are not available while the Properties dialog box is open. When the Find dialog box is opened from the Properties dialog box, closing the Properties dialog box also ends the Find command.

2. As needed, specify what you want to do with the Find operation. Click **Preferences**  to open the **Find Preferences** options dialog box.

To do this	Select this Option
Limit parts to include in the search.	Input Choose an option from the list.
Choose the action to apply to the search results.	Action • Show only (Loads matching parts) • Show additive (Loads matching parts)

To do this	Select this Option
	• Hide • Select only • Select additive • Deselect • Change appearance (Loads matching parts) You can click More Options to specify how the appearance of matched and unmatched results should change.
Include assembly nodes in the results, in addition to parts.	Include assembly nodes
Zoom the view window to the results of the search.	Zoom to matching results
Match only property values that have only the specified pattern characters.	Match whole word only
Match only property values where the character case matches the specified pattern character case.	Match case
Create a visual report of the results.	Click More Options to open the Visual Report Preferences dialog box and specify options for creating a visual report of the results.

3. In the **Find** dialog box, specify parameters for the property pattern you want to find.



For this pattern parameter	Do this
1) Property	Enter or select the property for the search pattern. Note: To refresh the list from properties of the currently selected nodes or parts, click Refresh Attributes List . If no nodes or parts were selected, a message appears prompting you to select a node or part.
2) Condition	Select the condition for comparing the property value. Note: For text values, only the = (contains) and != (doesn't contain) conditions are valid.
3) Value	Enter or select the value to search. The value can be either numbers or text. Note: When the value box (3) contains an asterisk (*), all parts and assemblies that have the selected property match the search pattern.

4. Click a **Find** command to find the matching occurrences and apply the specified actions.

- **Find Previous** — returns focus to a previous matching occurrence
- **Find Next** — sets focus to the next matching occurrence
- **Find All** — applies actions to all matching occurrences

Additional options

To do this	Click this command
Clear the applied action(s).	Clear  If no visual report was applied to the assembly, a message appears prompting you to reset the appearance to the default materials.

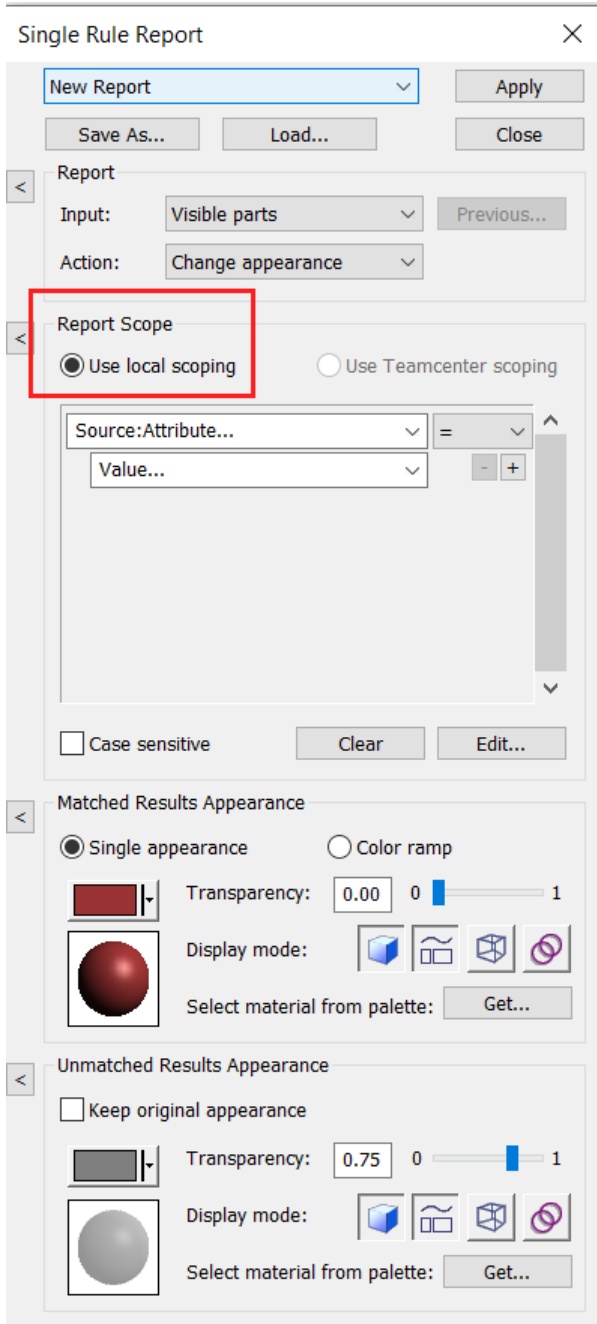
To do this	Click this command
	If a visual report was applied to the assembly, the appearance is reset to the state before the report was applied.
Select all matching parts found by the search.	Select Matching Parts 
Display an interactive legend for the search results.	Results Legend  For information about manipulating display of results in the Results Legend dialog box and in the Viewing window, see Examine results of the visual report .
Cycle through reports for properties or values (available in Windows OS only).	Explore Mode  and then: - Click either the property box or the value box to set focus on that parameter. - Press the up or down arrow key to choose the next item in the parameter list. Note: If focus is on the property box, when you press an arrow key to choose a new property, the value box automatically is set to asterisk (*) so that all values for the property are matched.
Refresh the properties and values lists.	Refresh Attributes List  Refreshes the lists from properties of the currently selected nodes or parts. If no nodes or parts were selected, a message appears prompting you to select a node or part.

Create a single rule visual report using local scoping

When your data is saved in the local file system you use local (client-side) scoping to generate visual report rules. The single rule report is used to search the parts of assemblies by composing a query for specific attributes and values.

- Open a 3D assembly model.
- On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .

3. (Optional) To create a new visual report using a previously saved report as a template, click **Load** and open the saved report.
4. In the **Single Rule Report** dialog box, in the **Report Scope** section, click **Use local** scoping.



5. In the **Report** section, specify the **Input** (parts to analyze) and **Action** (what to do to).
 - a. Do one of the following:

To analyze	Do this
All visible parts	From the Input list, choose Visible parts .
All selected parts	From the Input list, choose Selected parts .
All parts	From the Input list, choose All parts .
Matched occurrences from a previously applied report	Do the following: - Click Previous. - In the Previous Report dialog box, check the name of the report, the rule(s), and rule scope. (Optional) In the Matched Results section, click Select Parts to highlight parts that match the scope of the rule. For easy navigation, matched parts are also highlighted in the Assembly view. (Optional) In the Matched Results section, click Show Parts to only display the parts that match the scope of this rule. (Optional) Use the parts identified by either Select Parts or Show Parts to generate a second report that searches additional details and creates a visual report consistent with the new rules. Example: You need to provide two visual reports. One report will search for all parts release on a certain date (or relative date like [Today]-3); the second report displays designers who are overdue with their designs. You set the report scope for the first condition, all parts released on a certain date, and generate the visual report. Then set the report scope for the second condition, designers who are overdue, and then click Previous. Choose to apply the report scope of the second report to either Select Parts or Show Parts.

b. From the **Action** list, select one of the following:

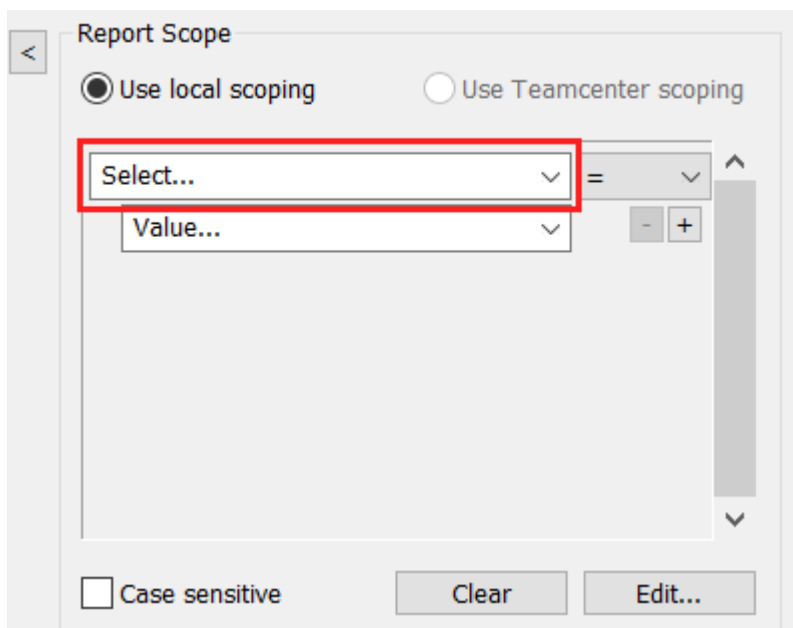
- **Change appearance**
- **Show only**
- **Show additive**

- **Hide**
- **Select only**
- **Select additive**
- **Deselect**

6. In the **Report Scope** section, specify the attribute, comparison operator, and value.

For example: Material = Steel or SignOff_Date > 2015/01/01

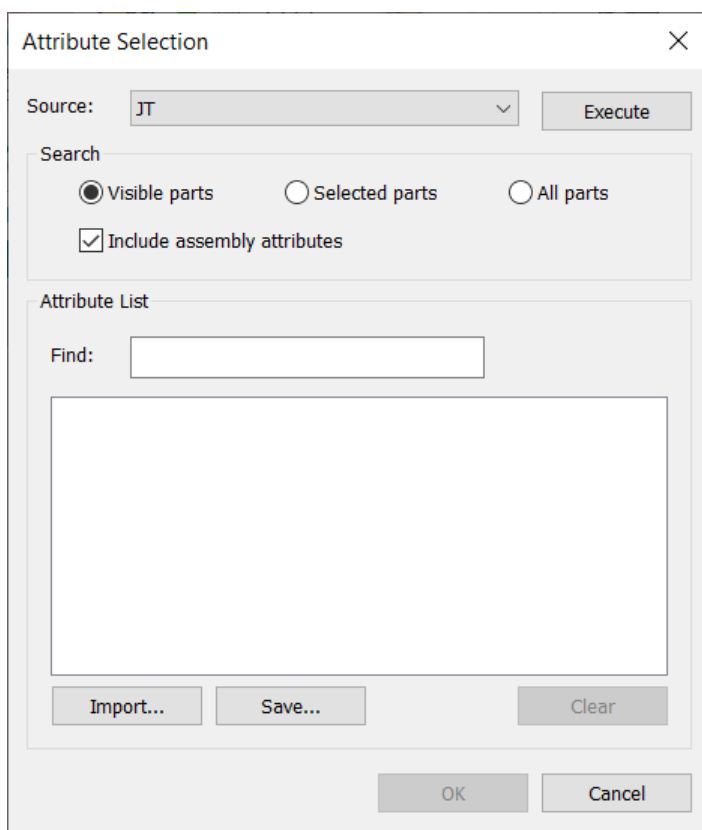
a. In the **Source:Attribute** box, type the attribute name or choose **Select**.



The screenshot shows the 'Report Scope' dialog box. At the top, there are two radio buttons: 'Use local scoping' (selected) and 'Use Teamcenter scoping'. Below these, there is a list box containing 'Select...' and 'Value...'. The 'Select...' option is highlighted with a red rectangle. To the right of the list box is a comparison operator dropdown menu showing '=', and below it are '-' and '+' buttons. At the bottom of the dialog, there is a 'Case sensitive' checkbox (unchecked) and two buttons: 'Clear' and 'Edit...'.

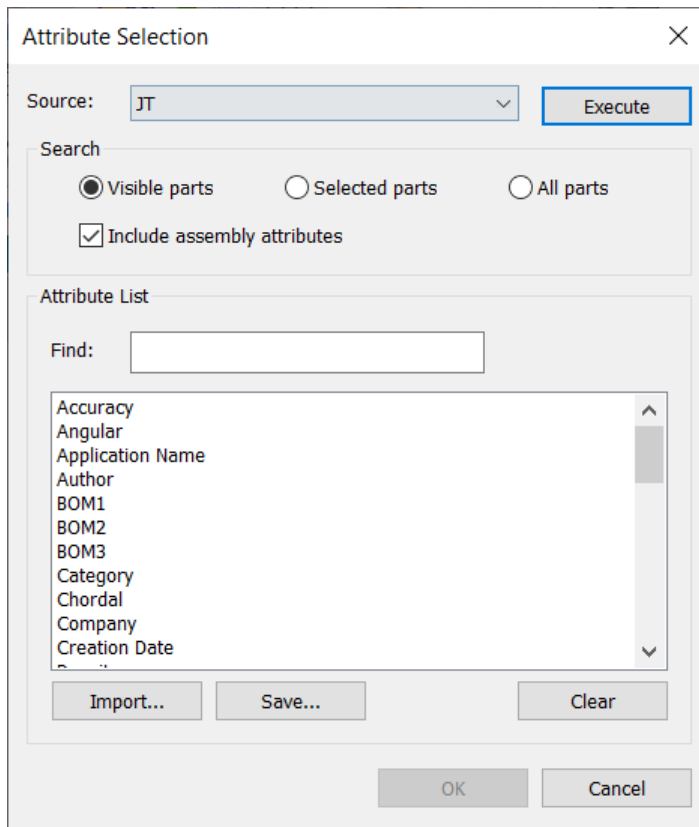
If you choose **Select**, the **Attribute Selection** dialog box appears.

b. Make sure **Source** is set to **JT**.



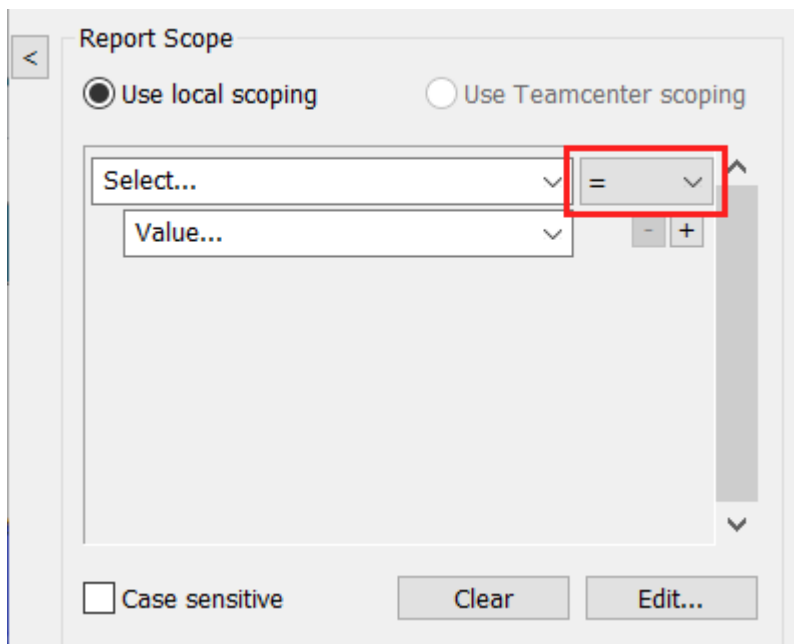
The image shows a software dialog box titled "Attribute Selection" with a close button (X) in the top right corner. The dialog is organized into several sections. At the top, there is a "Source:" label followed by a dropdown menu showing "JT" and an "Execute" button. Below this is a "Search" section containing three radio buttons: "Visible parts" (which is selected), "Selected parts", and "All parts". There is also a checked checkbox labeled "Include assembly attributes". The main part of the dialog is the "Attribute List" section, which includes a "Find:" text input field and a large, empty rectangular list box. At the bottom of the "Attribute List" section are three buttons: "Import...", "Save...", and "Clear". At the very bottom of the dialog are "OK" and "Cancel" buttons.

- c. Click **Execute** to generate a list of attributes for the assembly model.



The **Attribute Selection** dialog box is shown. It has a close button (X) in the top right corner. The **Source:** dropdown is set to **JT**, and the **Execute** button is highlighted with a blue border. Below this is a **Search** section with three radio buttons: **Visible parts** (selected), **Selected parts**, and **All parts**. There is also a checked checkbox for **Include assembly attributes**. The **Attribute List** section contains a **Find:** text box and a list of attributes: Accuracy, Angular, Application Name, Author, BOM1, BOM2, BOM3, Category, Chordal, Company, and Creation Date. At the bottom of the list are **Import...**, **Save...**, and **Clear** buttons. At the very bottom of the dialog are **OK** and **Cancel** buttons.

- d. Highlight the attribute and click **OK**.
- e. From the comparison operator list, select the operator.

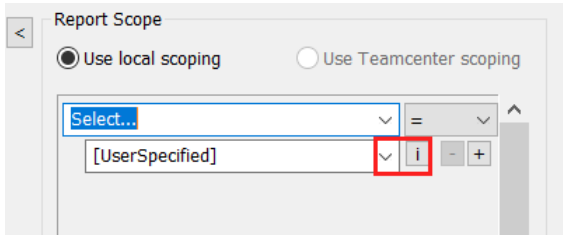
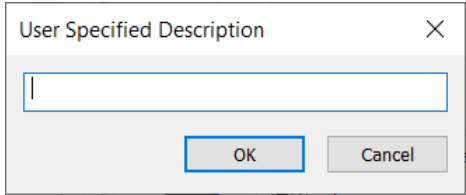


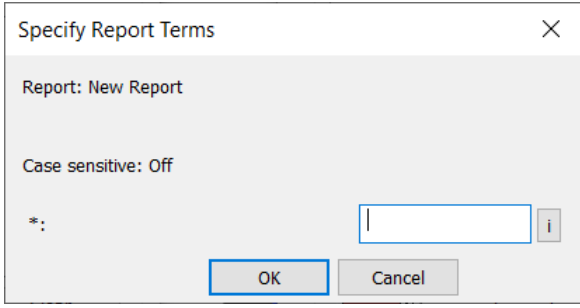
The **Report Scope** dialog box is shown. It has a back button (<) in the top left corner. There are two radio buttons: **Use local scoping** (selected) and **Use Teamcenter scoping**. Below these are two input fields: **Select...** and **Value...**, both with dropdown arrows. Between these fields is a comparison operator dropdown menu, which is highlighted with a red rectangle and shows the **=** operator. To the right of the **Value...** field are **-** and **+** buttons. At the bottom left is a **Case sensitive** checkbox. At the bottom right are **Clear** and **Edit...** buttons.

Some operators apply only to numeric or date attributes.

- f. Do one of the following to specify the value:

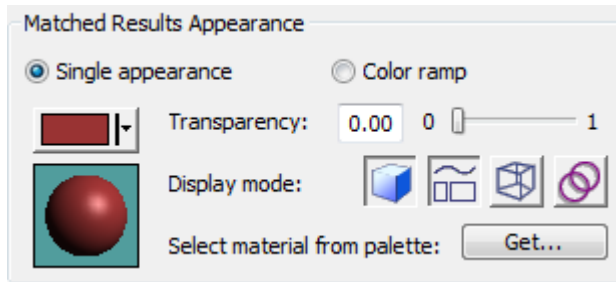
To	Do this										
Manually specify the value	Type the value in the Value box.										
Select the value from a generated list of values	A. Do the following: <table> <tr> <th>To display values for</th><th>Do this</th></tr> <tr> <td>One or more parts</td><td> i. Highlight the parts. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate. </td></tr> <tr> <td>One or more subassemblies</td><td> i. Highlight the subassemblies. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate. </td></tr> <tr> <td>One or more assemblies and all of their subassemblies and parts</td><td> i. Highlight the assemblies. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate. </td></tr> <tr> <td>All parts and subassemblies in a model</td><td> i. Highlight the model node. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate. </td></tr> </table>	To display values for	Do this	One or more parts	i. Highlight the parts. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate.	One or more subassemblies	i. Highlight the subassemblies. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate.	One or more assemblies and all of their subassemblies and parts	i. Highlight the assemblies. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate.	All parts and subassemblies in a model	i. Highlight the model node. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate.
To display values for	Do this										
One or more parts	i. Highlight the parts. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate.										
One or more subassemblies	i. Highlight the subassemblies. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate.										
One or more assemblies and all of their subassemblies and parts	i. Highlight the assemblies. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate.										
All parts and subassemblies in a model	i. Highlight the model node. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate.										

To	Do this
	B. Select the attribute value and click OK.
Specify a relative date	From the Value list, select one of the following: • [Today] • [Yesterday] • [PastWeek] • [PastMonth] • [Tomorrow] • [NextWeek] • [NextMonth] (Optional) Type + or – and an integer to add or subtract days. For example, specify [Today]-3.
Have the system prompt you to enter a value when you apply the report	Do the following: A. From the Value list, choose [UserSpecified]. B. (Optional) To enter a description of the attribute, click [i] and type the description in the User Specified Description dialog box.   When you apply the visual report, the Specify Report Terms dialog box appears and you can type the attribute value in the text box.

To	Do this
	 You can click [i] to display the attribute description.

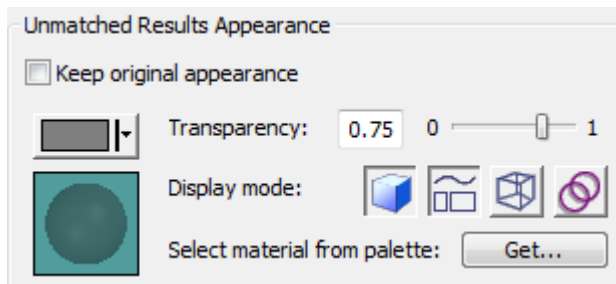
- g. (Optional) Select the **Case sensitive** check box to create string searches that are case sensitive.
- h. (Optional) Click **Edit** to **manually enter** the attribute, operator, and value string.
7. (Optional) Click + in the **Report Scope** section to specify an AND or OR attribute, operator, and value string.
8. (If the report **Action** is **Change Appearance**) In the **Matched Results Appearance** section, specify how you want to display parts that match your visual report scope.
 - a. Select a color pattern by doing one of the following:
 - To specify a single color for matched parts, select **Single appearance** and choose a color from the list.
 - To specify a color range, select **Color ramp**.
 - b. (Optional) If you specify a single color, in **Transparency**, move the slider to specify how much color transparency you want.
 - c. From **Display mode**, click one or more display styles.
 - d. (Optional) Click **Get** to select a material from the Materials palette.

This allows you to use advanced materials with **textures**.



9. (If the report **Action** is **Change Appearance**) In the **Unmatched Results Appearance** section, specify how you want to display parts that do not match your visual report query.
 - a. Select **Keep original appearance** to display unmatched parts in their original color.
 - b. Select a color from the list and set the **Transparency**.
 - c. From **Display mode**, click one or more display styles.
 - d. (Optional) Click **Get** to select a material from the Materials palette.

This allows you to use advanced materials with **textures**.

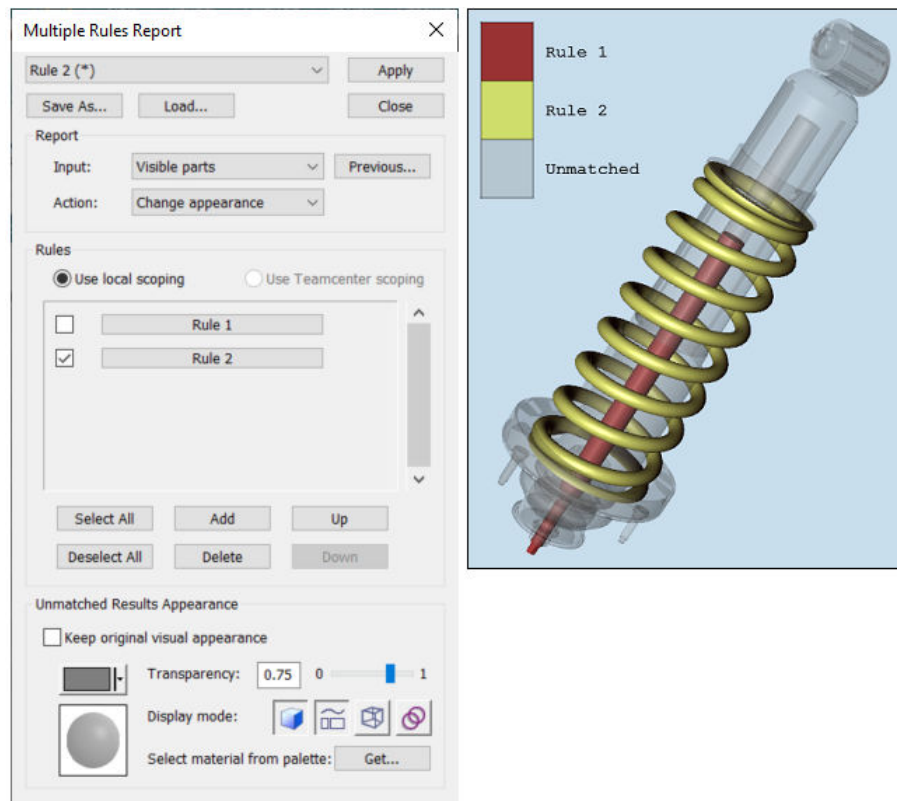


10. Click **Save As** or **Apply**.

Create a visual report containing multiple rules using local scoping

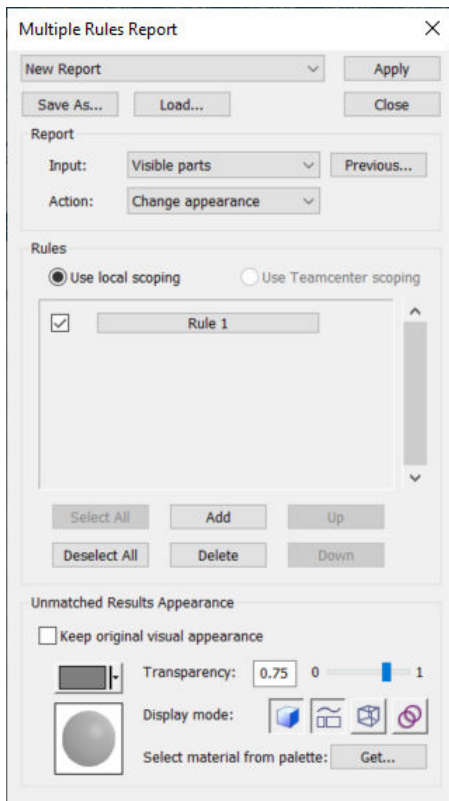
Use multiple rules reports when you need to query your assemblies with several rule constraints. When you use multiple rules to generate visual reports, the process is nearly the same as it is for generating single rule reports. You add several single rules using the **Single Rule Editor** dialog box that includes the **Report Scope** and **Matched Results Appearance** sections of the **Single Rule Report**.

Example:



1. On the **Visual Report** toolbar, click **Visual Report** , and then click **Multiple Rules Report** .

The **Multiple Rules Report** dialog box appears.



2. (Optional) To create a new visual report using a previously saved report as a template, click **Load** and open the saved report.
3. In the **Report** section, do the following:
 - a. Do one of the following:

To analyze	Do this
All visible parts	From the Input list, choose Visible parts .
All selected parts	From the Input list, choose Selected parts .
All parts	From the Input list, choose All parts .
Matched occurrences from a previously applied report	Do the following: A. Click Previous. B. In the Previous Report dialog box, check the name of the report, the rule(s), and rule scope. C. (Optional) In the Matched Results section, click Select Parts to highlight parts that match the scope of the rule.

To analyze	Do this
	For easy navigation, matched parts are also highlighted in the Assembly view. D. (Optional) In the Matched Results section, click Show Parts to only display the parts that match the scope of this rule. E. (Optional) Use the parts identified by either Select Parts or Show Parts to generate a second report that searches additional details and creates a visual report consistent with the new rules. Example: You need to provide two visual reports. One report will search for all parts release on a certain date (or relative date like [Today]-3); the second report displays designers who are overdue with their designs. You set the report scope for the first condition, all parts released on a certain date, and generate the visual report. Then set the report scope for the second condition, designers who are overdue, and then click Previous. Choose to apply the report scope of the second report to either Select Parts or Show Parts.

- b. In **Action**, select how you want to display the matched occurrences.

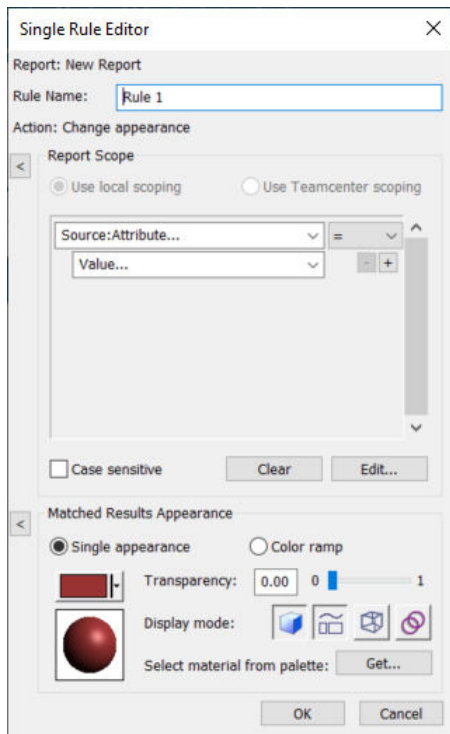
Note:

Only the **Change appearance** option supports editing **Matched Results Appearance** and **Unmatched Results Appearance**.

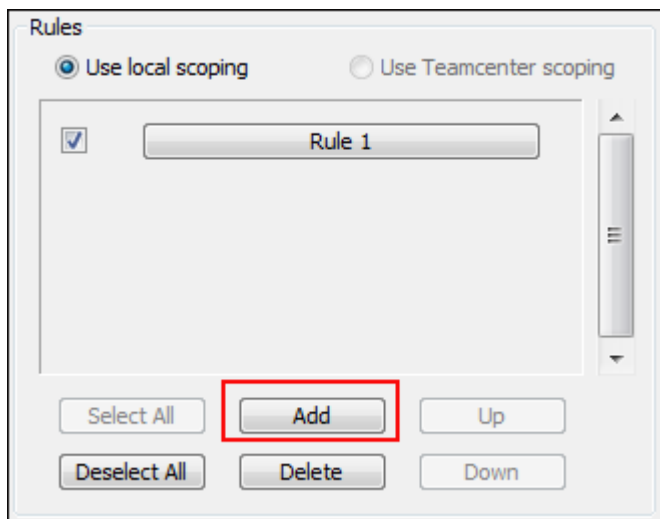
4. In the **Rules** section, create or modify the multiple rules for this report.

- a. Click **Rule 1**.

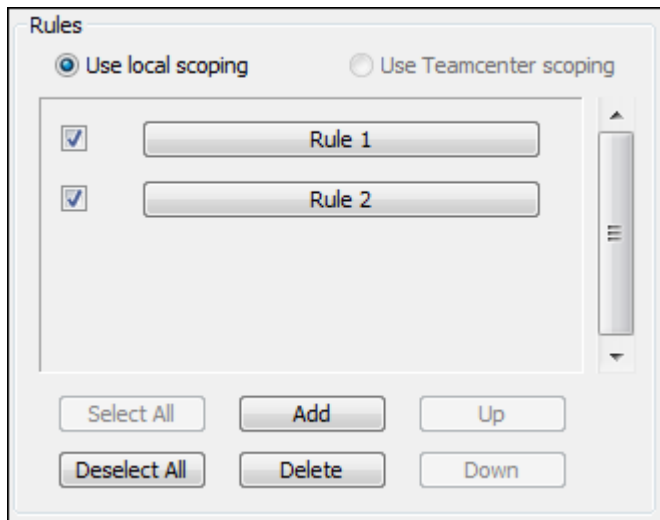
The **Single Rule Editor** dialog box appears.



- b. Configure the scope for the first rule exactly as you configure a **single rule report**, and then click **OK**.
- c. Click **Add** to add a second rule.



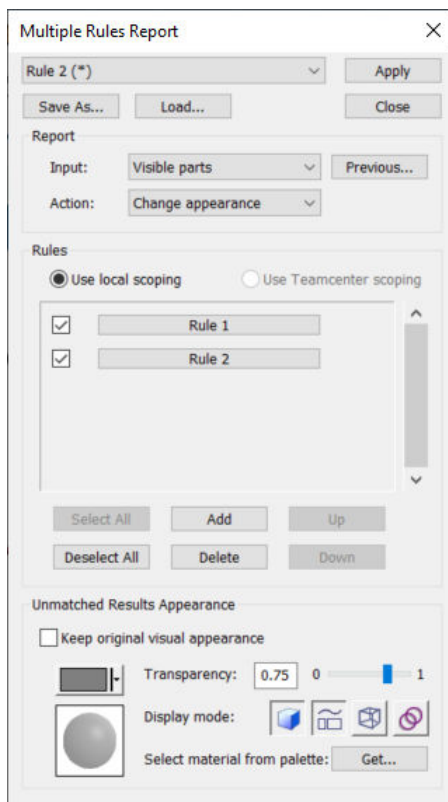
Rule 2 appears.



- d. Click **Rule 2**.

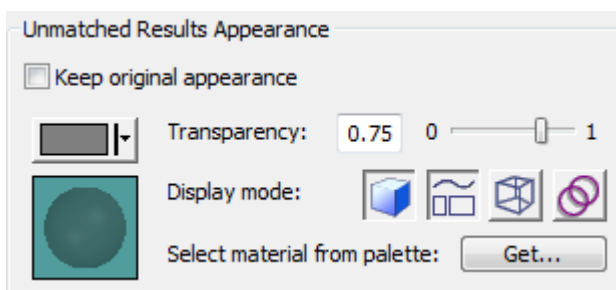
The **Single Rule Editor** appears again.

- e. Configure the scope for the second rule and click **OK**.



5. (Optional) In the **Rules** section, you can modify the rules (select, delete, and move) clicking any of the following:

- **Select All**
 - **Deselect All**
 - **Delete**
 - **Up**
 - **Down**
6. In the **Unmatched Results Appearance** section, specify how you want to display parts that do not match your visual report query.



- a. Select **Keep original appearance** to display unmatched parts in their initial color.
- b. Clear **Keep original appearance** and select a single color from the list.
- c. (Optional) In **Transparency**, move the slider to specify how much color transparency you want displayed.
- d. From **Display mode**, click one or more display styles.

You can test each mode or combination of modes. Click one or more display modes and then click **Apply**. The results are immediately displayed in the Viewing window.

- e. (Optional) Click **Get** to open the **Materials**, and then select a material to quickly display parts in a color of your choice.

Note:

For large assemblies, a visual report may display numerous colors for matched and unmatched parts. These colors may blend so closely that the results could be misinterpreted. In this situation, it may be useful to specify a known unmatched material color before you apply the report.

7. Click **Save As** or **Apply**.

Create visual reports using Teamcenter scoping

You can use Teamcenter scoping for visual reports when the assembly is loaded into the viewer from Teamcenter. Teamcenter scoping utilizes the Teamcenter cacheless search capabilities to provide access to attributes and values directly from the Teamcenter server.

Note:

The following describes using Teamcenter scoping for Single Rule Reports, but the process is the same process for Multiple Rule Reports.

1. Open the assembly in Teamcenter and select either **Start\Open in Lifecycle Visualization**, or right-click the assembly and choose **Send To→Lifecycle Viewer**.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then **Single Rule Report** .

In the **Single Rule Support** dialog box, **Use Teamcenter scoping** is automatically selected when the data is from Teamcenter.

Single Rule Report

New Report ▾ Apply

Save As... Load... Close

Report

Input: All parts ▾ Previous...

Action: Change appearance ▾

Report Scope

☐ Use local scoping ☒ Use Teamcenter scoping

Source: Attribute... ▾ = ▾

Value... ▾ - +

☐ Case sensitive Clear Edit...

Matched Results Appearance

☒ Single appearance ☐ Color ramp

Transparency: 0.00 0 1

Display mode:

Select material from palette: Get...

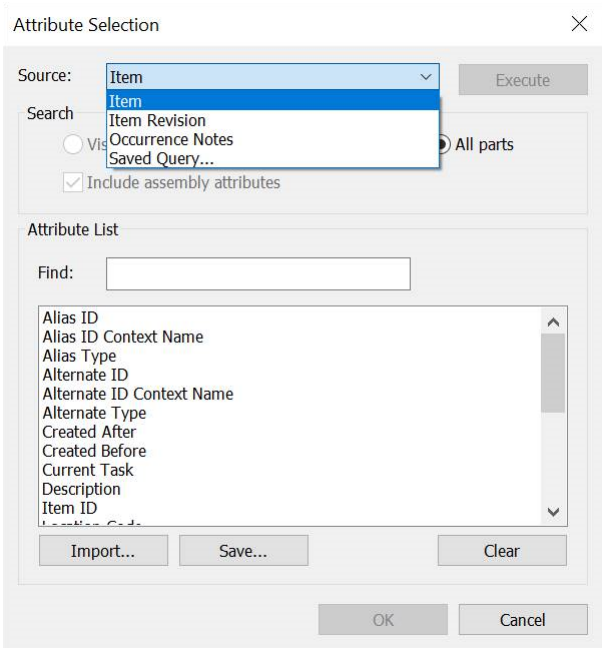
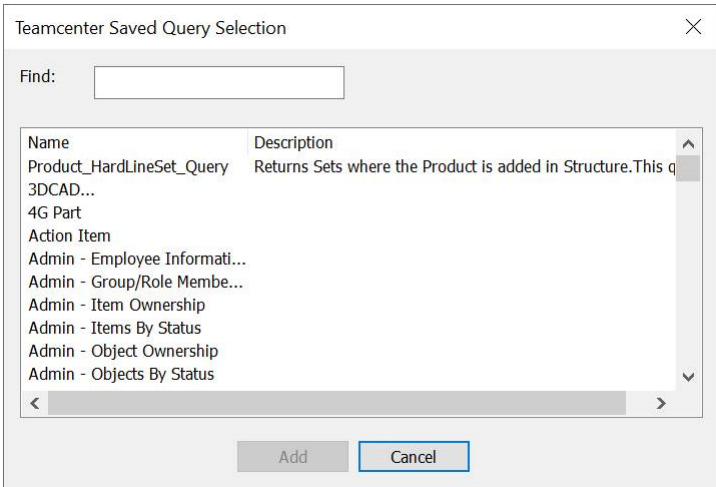
Unmatched Results Appearance

Note:

Teamcenter scoping is not supported with Workset or 4GD workset data.

3. Select **Source:Attribute**.
4. In the **Attribute Selection** dialog box, do one of the following:

To	Do this
Create a new query	a. From the Source list, select Item , Item Revision , or Occurrence Notes .

To	Do this
	 b. Select an attribute and click OK.
Use a save query	a. From the Source list, select Saved Query b. In the Teamcenter Saved Query Selection dialog box, select from the list queries that have been saved to the server. 

5. For details about configuring a **Single Rule Report**, see [Create a single rule visual report using local scoping](#).

Note:

For Teamcenter scoping, only AND is supported.

6. For details about configuring a **Multiple Rule Report**, see [Create a visual report containing multiple rules using local scoping](#)
7. When you are finished, click **Apply**.

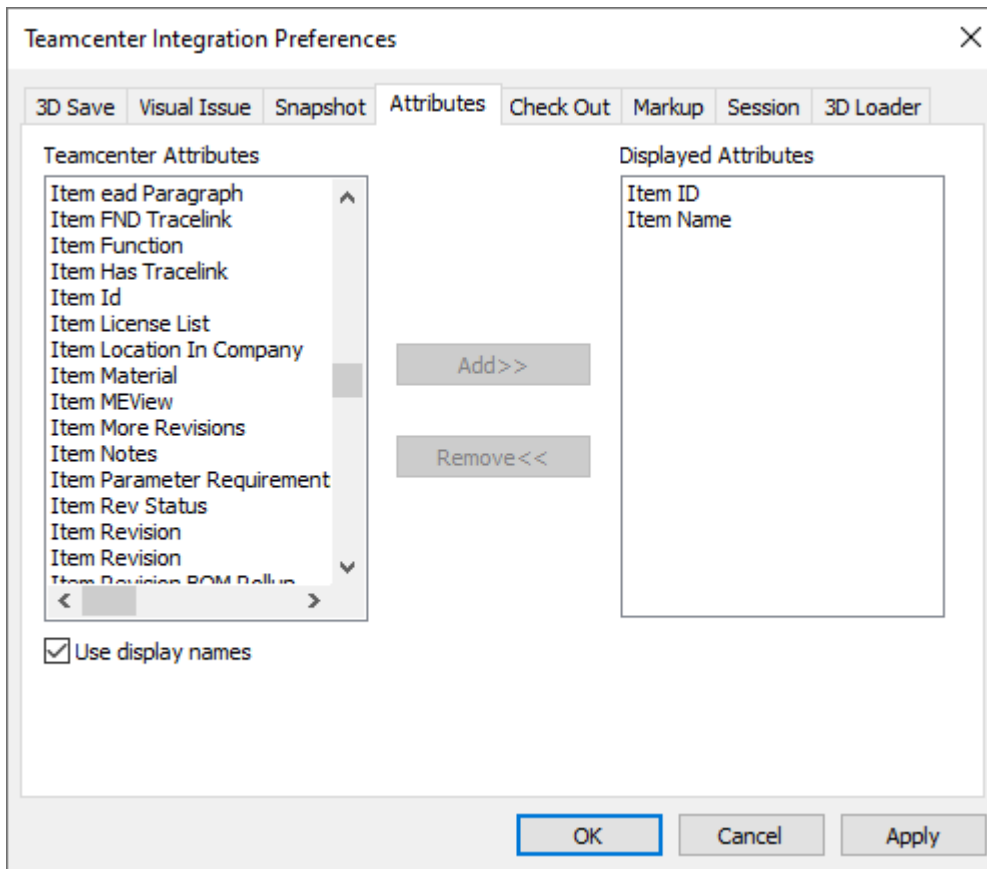
The results of your visual report are displayed in the Viewing window.

Create visual reports using Teamcenter structure lines

You can create client-side visual reports using Teamcenter structure cached on your local machine. This enables you to create visual reports using data not supported with Teamcenter scoping (cacheless searches), such as Worksets or 4GD worksets.

Prior to creating a visual report, you may need to specify the BOM line attributes to be loaded. If you are interested in 4G Designer element attributes, you do not need to specify the attributes to be loaded.

1. Send an object associated with 3D visualization data into the standalone viewer or the Lifecycle Viewer.
2. In the viewer, choose **File** tab→**Preferences**→**Teamcenter Integration**.
3. In the **Teamcenter Integration Preferences** dialog box that appears, select the **Attributes** tab.



4. In the **Teamcenter Attributes** section, highlight the attributes you want to include in the visual report, and then click **Add**.

Selected attribute objects appear in the **Displayed Attribute** section.

5. Click **OK**.
6. Close and reopen the file for the preference change to take effect.
7. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then **Single Rule Report**  or **Multiple Rules Report** .
8. In the **Report Scope** section, make sure **Use local scoping** is selected.
9. Select **Source:Attribute**, and in the **Attribute Selection** dialog box that appears, in **Source** select **Teamcenter Structure Lines**.

Notice the objects selected in **Teamcenter Integration Preferences** also appear in the **Attribute List** section of the **Attribute Selection** dialog box.

10. Click **Execute**.

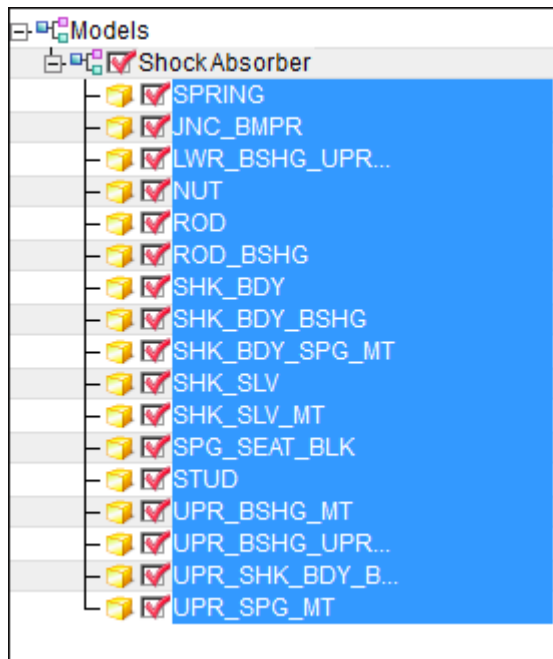
11. For details about configuring a **Single Rule Report**, see [Create a single rule visual report using local scoping](#).
12. For details about configuring a **Multiple Rule Report**, see [Create a visual report containing multiple rules using local scoping](#).

Create visual reports using relative dates

1. Open an assembly.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the Assembly tree, select at least one part.

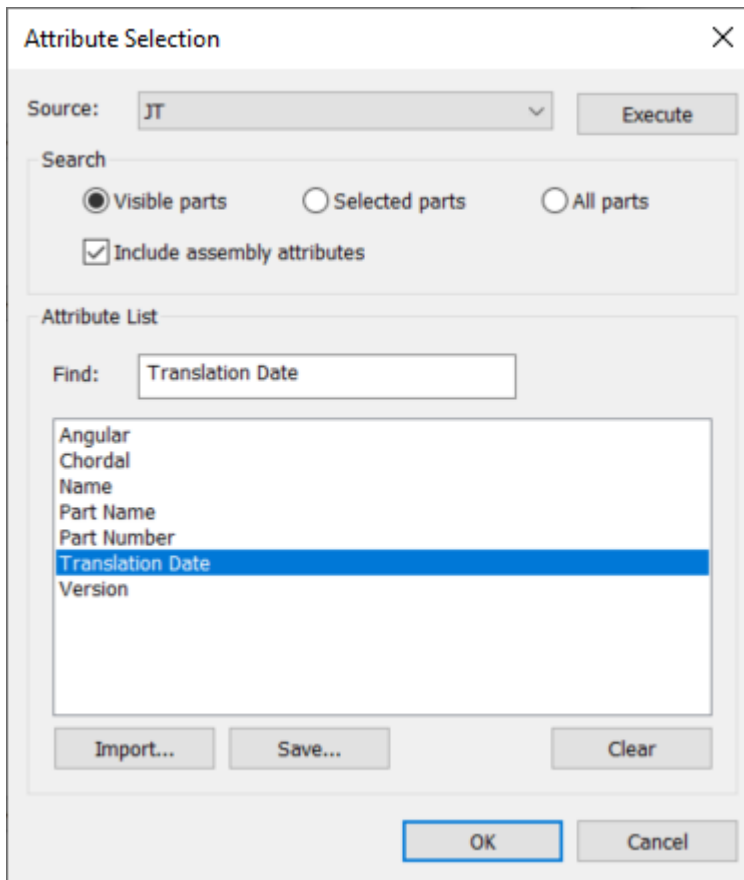
Note:

- If you select all of the parts, you display the entire list of values for this attribute.
- You must select at least one part on the Assembly view. If you do not highlight at least one part, an error message appears.
- If you select all the parts associated with the assembly, all available attribute values appear.



4. In the **Single Rule Report** dialog box, in the **Report Scope** section, choose **Source:Attribute** or **Select**.

5. In the **Attribute Selection** dialog box, to populate the **Attributes List** with the attributes of the selected parts, click **Selected Parts** and **Execute**.
6. Select a date attribute and click **OK**.



7. In the **Report Scope** section of the **Single Rule Report** dialog box, from the **Select** list, choose a condition from the following options:
 - [Today]
 - [Yesterday]
 - [PastWeek]
 - [PastMonth]
 - [Tomorrow]
 - [NextWeek]
 - [NextMonth]

8. (Optional) Type + or – and an integer to add or subtract days. For example, type [Today]-3.
9. In the **Single Rule Report** dialog box, click **Apply**.

If the condition evaluates as specified, the results are displayed in the Viewing window.

10. (Optional) Save the report to compare an instance of this condition in the future.

Comparing visual reports using snapshots

Compare visual reports using snapshots

You can compare the report scope of visual reports over time. For example, you can compare relative dates, version status, and completion status from one point in time to another point in time.

1. Open an assembly.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the **Single Rule Report** dialog box, in the **Report Scope** section, click **Source:Attribute** and then choose **Select**.
4. In the **Attribute Selection** dialog box that appears, click **Execute**, and select an attribute from the list.
5. Click **OK**.
6. Highlight at least one part in the Assembly tree.
7. In the **Single Rule Report** dialog box, in the **Report Scope** section, choose a value from the **Select** list.
8. Click **Apply**.
9. Save the visual report.
10. From the viewer, open the **Snapshots** view and select **Add**.
11. (Optional) In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Reset** .

This step resets the assembly to its original state.

12. Choose **File**→**Save Session As**, and save the session file.

13. At a later time, open the previously saved session file and load the saved visual report.
14. Follow previous steps to create another snapshot and save the session file again.
15. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
16. In the **Single Rule Report** dialog box, double-click one of the snapshots. Then, double-click the other snapshot.

The visual report results are displayed in the Viewing window. The corresponding visual **Report Scope** appears in the **Single Rule Report** dialog box.

Tip:

See an **example** about comparing visual reports using snapshots.

Example – comparing visual reports

This example creates snapshots of parts with different *version* numbers. The sample file is *shock_absorber.jt*.

You can use this example as a foundation for creating meaningful snapshots and visual reports. For example, you may want to compare when part design is completed, what the status is for each part during each design phase, and so forth.

1. Open Teamcenter lifecycle visualization and load *shock_absorber.jt*. This data is in the *Examples* folder, which is located in the Teamcenter lifecycle visualization installation directory.
2. On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the **Single Rule Report** dialog box, set **Report Scope** as (Version = 6).
4. In **Matched Results Appearance**, set the **Single appearance** to green.
5. Click **Apply**.
6. Navigate to the **Snapshots** view and click **Add**.

Note:

You may want to rename the snapshot, for example, *V6 green*.

7. Save this visual report, for example, *test_green.vpx*.

8. Save this session file as *shock_absorber_version6.vf*.
9. Follow the steps above to create and save a second visual report, snapshot, and session file.
10. For the second visual report, in **Report Scope**, type (Version = 2).
11. In **Matched Results Appearance**, set the **Single appearance** to red.
12. Click **Apply**.
13. Navigate to the **Snapshots** view and click **Add**.

Note:

You may want to rename the snapshot, for example, *V2 red*.

14. Save this visual report, for example, *test_red.vpx*.
15. Save this session file as *shock_absorber_version2.vf*.
16. Navigate to the **Snapshots** view and double-click one of the snapshots. Now double-click the other snapshot.

Both the Viewing window and the **Single Rule Report** dialog box display the selected visual report conditions. You can cycle from one snapshot to the other and review the different conditions.

You can cycle from one snapshot to another and review the scope of the visual report. If you created a visual report snapshot one month ago and then create one today, you can compare changes to the model.

3. Analyzing 3D models

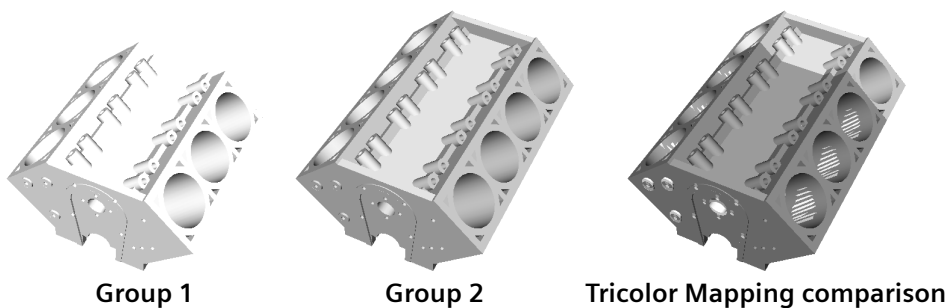
Comparing parts

Part comparison overview

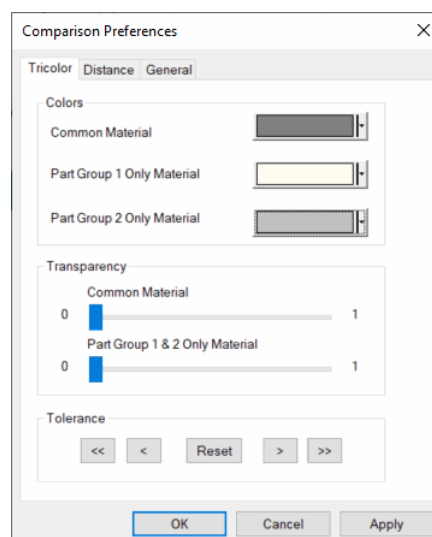
3D part comparisons help you to identify similarities and differences between part sets. There are two types of comparisons: geometric part comparisons in a single Viewing window and model comparisons in two or more windows displayed side by side.

Geometric part comparisons

Geometric part comparisons analyze two sets of parts that you specify, either from different models or from the same model, displaying the results of the comparison in a new Viewing window. Parts in group 1 are one color, parts in group 2 are a second color, and parts common to both groups are a third color.



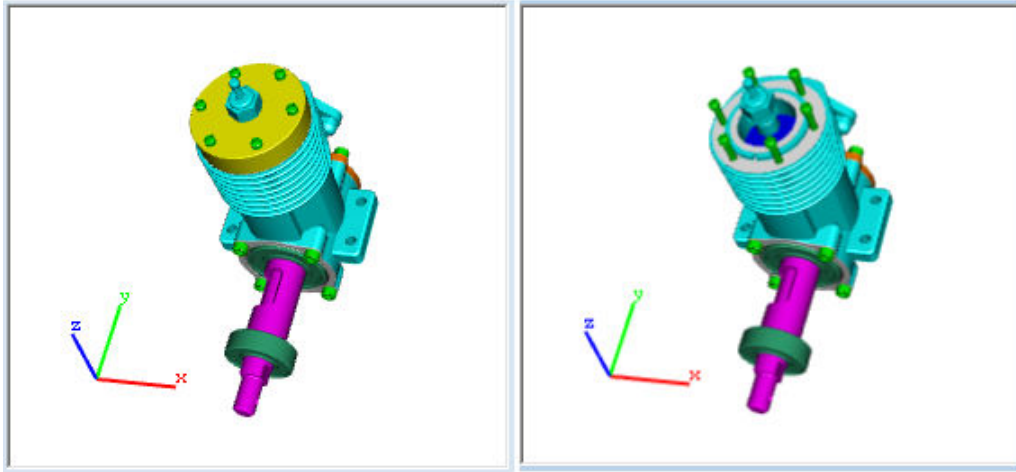
You can set color and tolerance preferences for geometric comparisons in the **Comparison Preferences** dialog box.



Comparison Preferences

Side by side window comparisons

Side by side window comparisons consist of two or more Viewing windows, which are tiled vertically or horizontally. You can synchronize the navigation cameras from the windows involved in the comparison.



Side by side window comparison

Compare 3D parts or models

Compare 3D model parts to identify the similarities and differences between corresponding parts or sets of parts.

1. Open two files and display the parts.
2. On the **3D Comparison** toolbar, click **Comparison** .
3. Select the parts to compare.
4. To create the part set for Group 1, on the **3D Comparison** toolbar, click **Set Part Group 1** .
5. In the second file in the Viewing window, select the parts to compare.
6. To create the part set for Group 2, on the **3D Comparison** toolbar, click **Set Part Group 2** .
7. (Optional) Set preferences for the appearance and tolerance of compared parts.
8. Create the compare view:

To identify	Do this
Similarities and differences	On the 3D Comparison toolbar, click Tricolor Mapping 
Extent of differences	On the 3D Comparison toolbar, click Distance Mapping 

In the **Comparison** Viewing window, differences and similarities of the parts are identified as you have specified.

Note:

- When you use cross section clipping with capping enabled and the **Intersecting Regions** option selected in clipping preferences, the common material of the tricolor mapping is also displayed in the intersecting regions capping color. However, the tricolor comparison colors are restored once clipping is removed or the section is deleted.
- If you perform a tricolor mapping comparison with some **Transparency** set in **Comparison Preferences**, transparency is correct only up to a maximum number of layers, which is influenced by both the graphics processing unit (GPU) in the system and the resolution of the display. For best results, a GPU supporting OpenGL 4.6 or later should be used, as it allows up to 17 correct layers when rendering at a resolution of 1920 x 1080 or less. Using higher resolutions results in fewer layers correct, with four layers being reached at a 4K resolution. Also, issues may result from older graphics drivers, therefore it is recommended that you update your graphics drivers the latest versions.

Tip:

- To minimize artifacts on the screen, if the models you are comparing have parts with multiple revisions, ensure that only one version of common parts is enabled in the models.
- GPUs capable of OpenGL 4.6 or later use an algorithm for tricolor compare that minimizes artifacts that may occur on the screen as you interact with the models. To use the algorithm:
 - a. On the **3D Performance** toolbar, click **Edit Performance Settings** .
 - b. On the **Rendering** tab, select **Advanced transparency** in both the **Moving Frames** and **Still Frames** sections.
 - c. Click **OK**.

9. (Optional) Select or display the parts.

To display the parts in this way	Do this
Select part set 1 in its Viewing window	On the 3D Comparison toolbar, click Show Part Group 1  .
Select part set 2 in its Viewing window	On the 3D Comparison toolbar, click Show Part Group 2  .
Display part set 1 in the Comparison Viewing window	In the Assembly , select the state indicator for Part Group 1 .
Display part set 2 in the Comparison Viewing window	In the Assembly , select the state indicator for Part Group 2 .

Perform side by side 3D comparisons

You can visually compare the contents of two or more different 3D Viewing windows.

1. Open and display a 3D model.
2. Open and display one or more additional models, or versions of the same model.
3. On the **3D Comparison** toolbar, click **Comparison** .
4. Click **Multiple window comparison** .

The Viewing windows are tiled vertically beside one another.

5. If you want to synchronize the navigation cameras in the Viewing windows, click **Synchronized Viewing** .
6. If you move or resize any of the Viewing windows and want to re-tile them, click **Reposition Windows** .

Specify part comparison preferences

You can change the appearances and tolerances for compared parts by specifying part comparison preferences.

The tricolor tolerance controls help you see how tight the tolerance is between the two part groups that are being compared. As you shift the tolerances, you can see how close the parts are to being common versus non-intersecting. The tolerance control allows users to shift one of the part group sets to exaggerate the intersecting geometry to give you quick visual comparison, but is not intended to show a quantitatively accurate view.

1. On the **3D Comparison** toolbar, click **Preferences** .
2. (Optional) Change appearances and tolerances in following ways:
 - For **Tricolor Mapping**, click the **Tricolor** tab, and then change the appearances and tolerance of compared parts using the following options:

To change	Take this action
Colors of parts that are common to both part sets	Click Common Material and choose a color.
Colors of parts that are in Part Group 1 only	Click Part Group 1 Only Material and choose a color.
Colors of parts that are in Part Group 2 only	Click Part Group 2 Only Material and choose a color.
Transparency of common parts	Drag the Transparency slider to the left (0) to make the parts more opaque, or to the right (1) to make the parts more transparent.
Tolerance (the distance allowed between common parts)	Do one of the following: • Click << or < to decrease the tolerance. • Click > or >> to increase the tolerance. • Click Reset

- For **DistanceMapping**, click the **Distance** tab, and then change the appearances and tolerance of compared parts using the following options:

To change	Take this action
Colors of parts that are common to both part sets	Click the Base Color button and choose a color.
Range of distances indicated by the color map	a. Select the Low Value check box and type the minimum distance to identify with the color map. Note: The specified minimum distance is mapped to the blue end of the color spectrum. b. Select the High Value check box and type the maximum distance to identify with the color map.

To change	Take this action
	Note: the specified maximum distance is mapped to the red end of the color spectrum. Note: - Low and high values are automatically converted to model units. For example, if the model unit is millimeters and you enter 10 in the High Value check box, 10 millimeters will be the maximum distance. - Distances outside the color range are mapped to your base color. You can select a base color that works best for your model. - If you do not set minimum or maximum distance values, after processing the minimum distance is mapped to the blue spectrum and the maximum distance is mapped to the red spectrum. Zero differences are mapped to your base color. Tip: To view calculated minimum and maximum distances after processing, navigate to the Distance tab of the Comparison Preferences dialog box and read the values in the Low and High Value sections.

- To change basic comparison preferences, click the **General** tab, and then change preferences in these ways:

To change	Take this action
Default comparison type	Select Tricolor or Distance Mapping .
Coordinates at which parts are displayed	Select or clear Use World Coordinates .

To change	Take this action
	Tip: Select Use World Coordinates to compare parts in an entire model. Clear Use World Coordinates to compare two versions of a single part.

- Click **Apply** or **OK**.

When you compare parts, the common parts appear in the base color. Parts that are different appear in the colors of the color map, with parts that are the low value distance apart colored blue, and parts that are the high value distance apart colored red.

Sectioning models

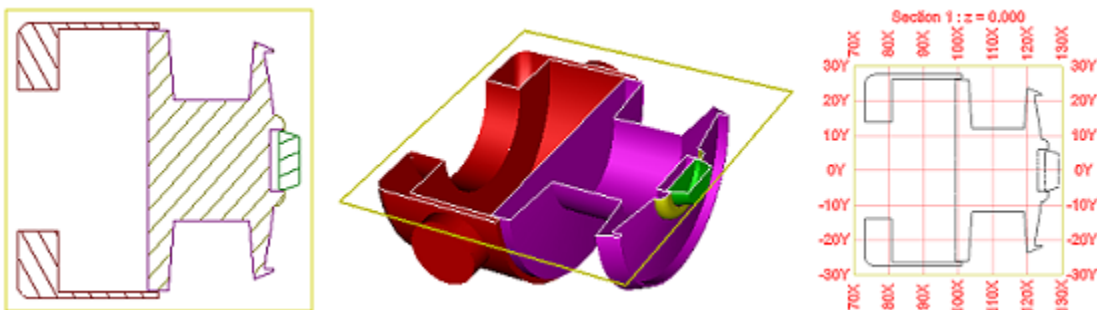
Sectioning models

You can manipulate a plane through your model and **display a cross section of the model** where it is cut by the plane. Section planes can be axis **aligned** or **positioned** along a curve. They can appear with or without grid lines and labels.

Note:

Parts that contain the property **TcVis_Concept_Background_Geom** are considered part of the background or environment and do not have cross sections applied to them.

Place the **TcVis_Concept_Background_Geom** property on both the instance and part nodes. Syntax is **TcVis_Concept_Background_Geom = True**.



Create a cross section

- On the **3D Section** toolbar, click **Enable Section Tools** .

2. (Optional) Select specific parts to section if you do not want the entire model sectioned.
3. Do one of the following:

To create a cross section	Do this
Aligned to an axis	Do one of the following: - On the 3D Section toolbar click Create Section . Note: When you use the 3D Section toolbar to create a section, the plane appears in the 3D Viewing window, normal to the Z axis if this is the first section you have created. For subsequent sections, the plane is normal to the axis of the last created section or the axis of the last section aligned using the Position Plane dialog, whichever happened most recently. - Choose Section3D→Create Section and select X, Y, or Z. - On the 3D Section toolbar click Create Grid . - Choose Section3D→Create Grid and select X, Y, or Z.
Along a curve Note: - You can only pick curves on models that contain either B-Rep data or wireframe data. - When you save a section created along a curve to a snapshot or session file, the curve is not persisted. - The User Defined Step Size preference is ignored for sections along curves. The value on the Position Plane slider ranges from 0 (start of curve) to 1 (end of curve).	a. Do one of the following: - Choose Section3D→Create Section→Along Curve. - On the 3D Section toolbar, click Create Section Along Curve . b. In the Viewing window, select a single curve or multiple connected curves on a part. The selected curve(s) are highlighted. c. In the Select Curve(s) dialog box, click Done.

To create a cross section	Do this
The slider step size is always 0.01 (1/100 of the distance normal to the plane in the visible scene).	

In Professional and Mockup, a cross section appears in the following places:

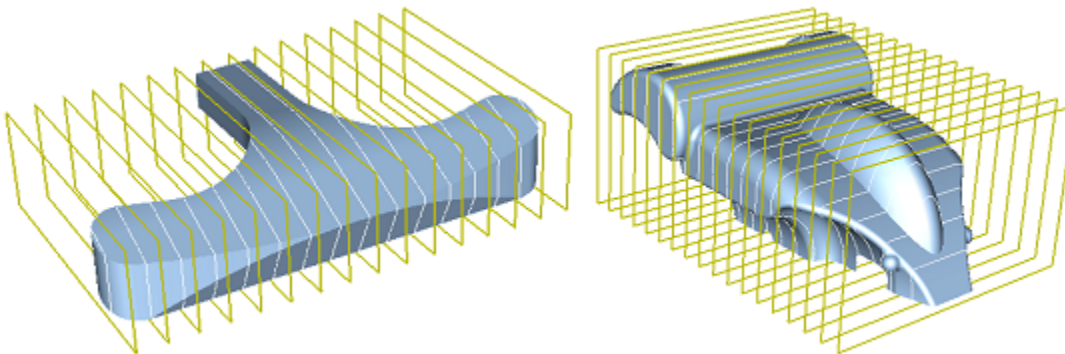
- The 3D Viewing window
 - The **Cross Sections** view
 - (If you select **Display Active Section in Viewer** in the cross section preferences) The Cross Section Viewer
4. (Optional) Use the **Position Plane** dialog box to change the position and axis of the section plane.
 5. (Optional) In Professional and Mockup, add or remove parts from the section.
 6. (Optional) **Delete** the cross section.

Tip:

You can show or hide the cross section border by choosing **Section3D→Show Borders** or **Hide Borders**.

Create a series of cross sections

You can create multiple cross section views along an axis, along a linear edge, or between two points.



1. On the **3D Section** toolbar, click **Enable Section Tools** .

2. (Optional) Select specific parts to section if you do not want the entire model sectioned.
3. On the **3D Section** toolbar, click **Create Series** .
4. In the **Create 3D Section Series** dialog box, do one of the following:

Choose one of the following	Then do this
Along Axis	Select X , Y , or Z .
Along Edge	• Click Pick Linear Edge. • In the Viewing window, pick an edge.
Between 2 Points	• Click Pick 2 Points. • In the Viewing window, click two points.

5. (If you want to specify the amount of space between planes) Select **Enter Spacing Between Planes**, and enter the amount.
6. (If you want to specify the number of planes created) Select **Enter Number of Planes**, and enter the number.

Note:

If you enter a value for **Enter Spacing Between Planes**, the **Enter Number of Planes** value automatically updates to match the amount. The resultant number is the number of planes that are necessary to fill the bounding box of the visible parts of your model using the specified spacing.

7. In the **Start Point** section, do one of the following:
 - Select **Enter**, and enter 3D coordinates.
 - Choose **Select**, click **Pick Start Point**, and then pick a point on your geometry.
8. Click **Apply** to create the cross section planes without closing the dialog box, or click **OK** to create the planes and close the dialog box.

In the Viewing window, the specified number planes appear, parallel and evenly spaced along the axis.

Delete cross sections

Do one of the following:

To	Do this
Delete a specific cross section	1. Select the section plane to delete. 2. Do one of the following: • On the 3D Section toolbar, click Delete Section . • In the Viewing window, right-click the plane, and select Delete.
Delete all cross sections	On the 3D Section toolbar, click Delete All  .

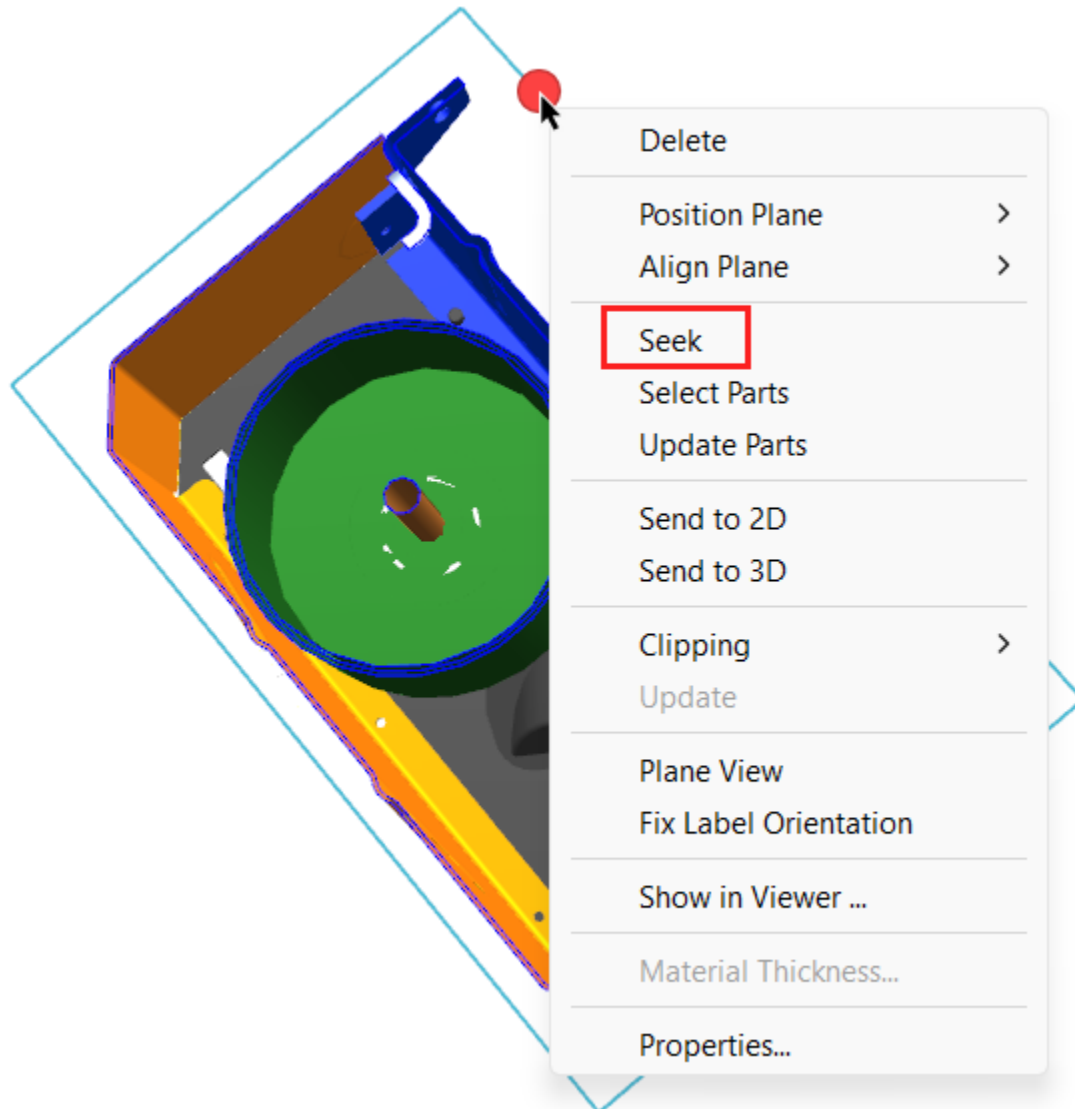
Note:

You can use the **Undo** command to undo the last cross section deletion action.

Center a point of the section

When you center a point of your section in the Viewing window, the point becomes the center of rotation.

- In the 3D Viewing window, do either of the following:
 - Click a point on the section plane using the middle mouse button (or wheel).
 - Right-click a point on the section plane and choose **Seek**.



Your view changes so that the point you clicked is at the center of the Viewing window.

Reposition section planes

Overview of repositioning section planes

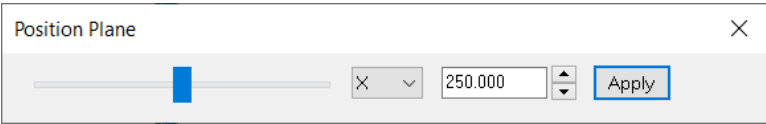
You have several options for changing the orientation and position of section planes. You can:

- Move the section plane along its normal and align it to an axis using the **Position Plane** dialog box.
- Dynamically move and rotate the section plane using a 3D manipulator.
- Dynamically move and rotate the sectioning plane using a 3DConnexion device.

- Align the section plane to an axis, the view, or specific model geometry.

Reposition section planes using the Position Plane dialog box

You can reposition section planes in your model using the **Position Plane** dialog box. You can align a section plane to an axis, move the plane along an axis (or curve in Professional or Mockup), or position the plane a precise distance from the origin.



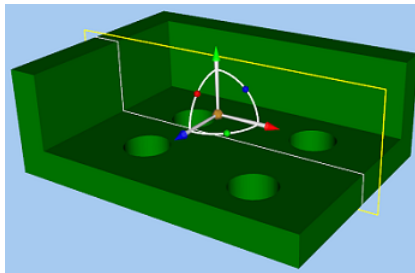
- Create a cross section.
- On the **3D Section** toolbar, click **Position Plane** .
- Do any of the following:

To	Do this
Align the plane to the X, Y, or Z axis	In the Position Plane dialog box, click in the center box and choose X, Y, or Z.
Move the plane along its normal	Do one of the following: - In the Position Plane dialog box, drag the slider left or right. - In the Position Plane dialog box, click to the left or right of the slider to move it ten steps to the left or right. - Press the up, down, left, or right arrow keys on your keyboard to move the slider one step to the left or right. - Press the page up or page down keys on your keyboard to move the slider ten steps to the left or right. - Press the home or end keys on your keyboard to move the slider all the way to the left or right. Tip: In the Position Plane dialog box, click the arrows up or down to precisely move the section by single increments of the step size.

To	Do this
	Note: By default, the Position Plane slider uses steps that are 1/100 of the total range of the slider, which is the distance normal to the plane in the visible scene. In Professional and Mockup, you can specify a custom step size when you set cross section preferences. However, the custom step size is ignored by sections along curves.
Position the plane a precise distance from the origin	In the Position Plane , type a value in the offset box and click Apply .

Reposition section planes using 3D manipulators

You can dynamically move and rotate section planes using a 3D manipulator.



Note:

Using a 3D Manipulator is not supported for Section Series and Section Along a Curve.

1. Select an existing section or create a new one.
2. If the **3D Manipulator** is not already displayed in the center of the section plane, do one of the following:
 - On the **3D Section** toolbar, click **3D Manipulator** .
 - In the Viewing window, right-click the section plane and choose **Position Plane→3D Manipulator**

The manipulator appears in the center of the section plane.

3. Do any of the following:

To	Do this						
Drag the section plane along its normal	Drag the blue manipulator axis. Note: The blue axis is always perpendicular to the section plane.						
Translate the section plane a precise delta distance along its normal	- If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. - Click the blue manipulator axis to display the distance window. - Do one of the following: <table> <tr> <th>To</th><th>Do this</th></tr> <tr> <td>Drag using a snap increment</td><td> (Optional) Type a new Snap value in the distance window. - Drag the axis. If Snap is set to 5, the plane move in 5 unit increments. </td></tr> <tr> <td>Specify a delta distance</td><td> Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value. </td></tr> </table>	To	Do this	Drag using a snap increment	(Optional) Type a new Snap value in the distance window. - Drag the axis. If Snap is set to 5, the plane move in 5 unit increments.	Specify a delta distance	Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.
To	Do this						
Drag using a snap increment	(Optional) Type a new Snap value in the distance window. - Drag the axis. If Snap is set to 5, the plane move in 5 unit increments.						
Specify a delta distance	Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.						
Translate the section plane by typing a new Z offset value	- If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. - Click the manipulator origin to display position window. - Enter the new Z value. (Optional) Enter new X and Y values to translate the manipulator on the section plane.						

To	Do this						
Rotate the section plane about a manipulator axis	Do one of the following: • Drag the red sphere to rotate the plane around the red manipulator axis. • Drag the green sphere to rotate the plane around the green manipulator axis.						
Rotate the section plane a specific number of degrees around the red or green manipulator axis	a. If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click the red or green sphere to display the rotation window. c. Do one of the following: <table border="1"> <thead> <tr> <th>To</th><th>Do this</th></tr> </thead> <tbody> <tr> <td>Drag using a snap increment</td><td> A. (Optional) Type a new Snap value in the rotation window. B. Drag the sphere. If Snap is set to 15, the plane rotates in 15 degree increments. </td></tr> <tr> <td>Specify a delta angle</td><td> Enter the Angle in the rotation window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value. </td></tr> </tbody> </table>	To	Do this	Drag using a snap increment	A. (Optional) Type a new Snap value in the rotation window. B. Drag the sphere. If Snap is set to 15, the plane rotates in 15 degree increments.	Specify a delta angle	Enter the Angle in the rotation window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value.
To	Do this						
Drag using a snap increment	A. (Optional) Type a new Snap value in the rotation window. B. Drag the sphere. If Snap is set to 15, the plane rotates in 15 degree increments.						
Specify a delta angle	Enter the Angle in the rotation window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value.						
Rotate the manipulator to change the directions of the red and green axes	Drag the blue sphere to rotate the manipulator and the two rotation axes.						
Move the axis of rotation away from the center of the plane.	Do one of the following: • Click the red or green manipulator axis and translate the manipulator. • Click the manipulator origin and drag the manipulator.						
Return the manipulator to the center of the section plane	Right-click the manipulator and choose Snap to Section Center .						

To	Do this						
Align the section plane to coordinate systems	Do one of the following: <table> <tr> <th>To</th><th>Do this</th></tr> <tr> <td>Select a coordinate system in the Viewing window</td><td> a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window. </td></tr> <tr> <td>Select a coordinate system from a list</td><td> a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK. </td></tr> </table> The section plane moves and rotates to align to the coordinate system.	To	Do this	Select a coordinate system in the Viewing window	a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window.	Select a coordinate system from a list	a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK.
To	Do this						
Select a coordinate system in the Viewing window	a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window.						
Select a coordinate system from a list	a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK.						
Rotate the section plane by aligning a manipulator axis to a part feature	a. Right-click a manipulator axis and choose Align Axis. b. Select a part feature.						
Rotate the section plane by aligning a manipulator plane to a part feature	a. Right-click a manipulator ball and choose Align Plane. b. Select a part feature.						
Translate the section plane by aligning the manipulator origin to a part feature	a. Right-click the manipulator and choose Snap to Feature Geometry. b. Select a part feature.						

Reposition section planes using a 3DConnexion device

You can dynamically move and rotate the cross sectioning plane in the X, Y, Z, or arbitrary direction using a 3DConnexion device.

Out of the box, the 3DConnexion device moves the camera. Hold the Ctrl key to move the section plane. You can reverse the behavior by setting a preference in the **Sensor Navigation Preferences** dialog box.

You can move the plane in either the World Coordinate System or in Screen Coordinate System. Set the preference in the **6-DOF Input Device** section on the **General** tab of the **Cross Section Preferences** dialog box.

Align cross sections

Choose to align a section plane to an axis, the view, or specific model geometry.

Note:

These options do not apply to section planes created with Section Series and Section Along a Curve.

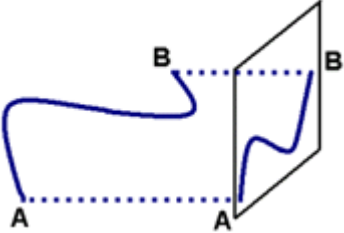
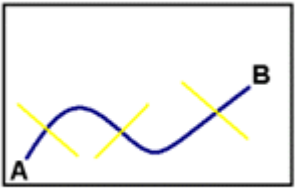
1. Create or select the cross section plane that you want to align.
2. Align the plane to a feature in one of these ways:

To align a plane	Do this
To an axis	Do one of the following: • In the Viewing window, right-click the section and choose Align Plane→To Axis, and then X, Y, or Z. • Choose Section3D→Align Plane→To Axis, and then X, Y, or Z. The plane is aligned so it is normal to the axis.
To the current view	Do one of the following: a. Do one of the following: • In the Viewing window, right-click the section and choose Align Plane→To View. • Choose Section3D→Align Plane→To View. b. Do one of the following: • Choose Horizontal to rotate the plane so it aligns with the X axis of your view. • Choose Vertical to rotate the plane so it aligns with the Y axis of your view. • Choose Perpendicular to rotate the plane so it is parallel to the viewing plane.
To a measurement	a. Do one of the following: • In the Viewing window, right-click the section and choose Align Plane→To Measurement. • Choose Section3D→Align Plane→To Measurement. • On the 3D Section toolbar, click Align to Feature .

To align a plane	Do this
	b. In the Viewing window, click a measurement label. The plane is aligned to the measurement.
Through a point	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Point. - Choose Section3D→Align Plane→To Point. - On the 3D Section toolbar, click Align to Feature . b. Do one of the following: - Select a point, vertex, arc center, midpoint, or point on edge on the model. Calculate a point. The plane passes through the selected point. It remains parallel to its original orientation.
To an edge	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Edge. - Choose Section3D→Align Plane→To Edge. - On the 3D Section toolbar, click Align to Feature . b. In the Viewing window, click an edge. If you select a straight edge, the plane is aligned so it is normal to the selected edge at its midpoint. If you select a curve, the plane is aligned to the plane of the curve.
To a surface	Note: When you align a cross section plane to a conic surface, the plane aligns with the axis of symmetry of the conic. a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Surface. - Choose Section3D→Align Plane→To Surface.

To align a plane	Do this
	On the 3D Section toolbar, click Align to Feature . b. In the Viewing window, click a surface. The plane is aligned so its normal vector matches the selected surface's normal vector.
To a part	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Part. - Choose Section3D→Align Plane→To Part. - On the 3D Section toolbar, click Align to Feature . b. In the Viewing window, click a part. The plane is aligned to the center of the selected part's bounding box, normal to the Z axis.
Between two parts	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→Between 2 Parts. - Choose Section3D→Align Plane→Between 2 Parts. b. In the Viewing window, click the two parts. The section is aligned so it contains the minimum distance line between the two parts.
Between two points	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→Between 2 Points. - Choose Section3D→Align Plane→Between 2 Points. b. In the Viewing window, click two points. The plane is aligned so it is centered between the two points and normal to a line through the two points.
Through three points	a. Do one of the following: In the Viewing window, right-click the section and choose Align Plane→Through 3 Points.

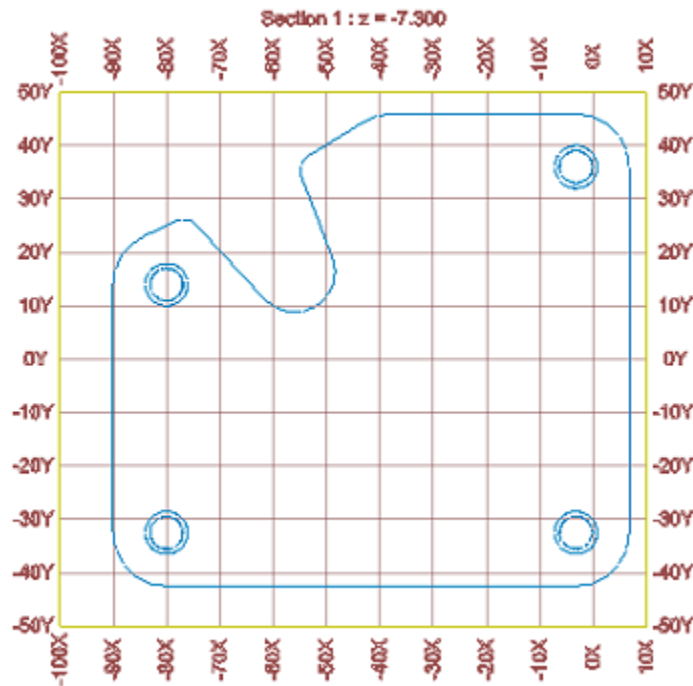
To align a plane	Do this
	Choose Section3D→Align Plane→Through 3 Points. b. In the Viewing window, click three points. The plane is aligned so it passes through all three points.
Normal to curve at a point	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Curve at Point. - Choose Section3D→Align Plane→To Curve at Point. b. In the Viewing window, click an edge on the 3D model. c. In the Align Plane Normal to Curve at Point dialog box, do one of the following: - Click Select Point, click OK, and then in the Viewing window, click a point or vertex on the edge. - Click Identify X, Y, or Z Coordinate on Curve, select an axis from the list, enter an offset value, and click OK. The section is aligned normal to the curve at the specified point.
To a projected curve at a point	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Projected Curve at Point. - Choose Section3D→Align Plane→To Projected Curve at Point. b. In the Viewing window, click an edge on the 3D model. c. In the Align Plane Normal to Curve at Point dialog box, do one of the following: - Click Select Point, click OK, and then in the Viewing window, click a point or vertex on the edge. - Click Identify X, Y, or Z Coordinate on Curve, select an axis from the list, enter an offset value, and click OK. The section is aligned to the 2D projection of the curve at the specified point. Note: Sections aligned to the 2D projection of an edge will always be displayed as an edge view. When you select To Projected Curve at Point the section is aligned to the 2D projection of the 3D curve at the picked point. The

To align a plane	Do this
	images below show how the sections will be displayed. Sections aligned to the 2D projection of a curve are always displayed as an edge view.  Projection of 3D curve to display surface  Sections cut normal to the 2D projection of the original 3D curve
To a surface at a point	a. Do one of the following: • In the Viewing window, right-click the section and choose Align Plane→To Surface at Point. • Choose Section3D→Align Plane→To Surface at Point. b. In the Viewing window, click the surface. c. Do one of the following: • Select a point, vertex, arc center, midpoint, or point on edge on the model. • Calculate a point. The plane passes through the point and is aligned so its normal vector matches the selected surface's normal vector.
To a coordinate system	a. Do one of the following: • In the Viewing window, right-click the section and choose Align Plane→To Edge. • Choose Section3D→Align Plane→To Edge. • On the 3D Section toolbar, click Align to Feature .

To align a plane	Do this
	b. In the Viewing window, click an axis of the coordinate system. The plane is aligned to the coordinate system, perpendicular to the selected axis.
To a PMI coordinate system	a. Do one of the following: - In the Viewing window, right-click the section and choose Align Plane→To Surface. - Choose Section3D→Align Plane→To Surface. - On the 3D Section toolbar, click Align to Feature . b. In the Viewing window, click a surface that represents a coordinate plane on the axis of the PMI local coordinate system. The plane is aligned to the coordinate system.

Overview of section grid options

Grid lines help you understand the size, orientation, and location of parts in section views. Grid line labels include the axis and location. Section titles include the section number, alignment axis, and location along the axis.



You specify cross section preferences to:

- Show or hide the grid.
- Change grid line spacing, color, width, and transparency.
- Display grid line labels and a section title.
- Change the text size and color of the labels and title.
- Have the labels and title update to fit the screen when you zoom and pan while in Plane View.

Note:

- Cross section grid lines are based on the global coordinate system.
- Only axis-aligned section planes can display grid lines.
- Section lines are turned off by default when you create a grid.

Edit material thickness values

Display a virtual line to represent the material thickness of parts. By creating an additional section line outline with a specific offset, the cut surface data appears to have thickness.

You can set global values for material thickness, and edit material thickness values after they are applied by setting cross section preferences.

1. Create a cross section.
2. Enable material thickness by setting cross section preferences.
3. In the Viewing window, select a part, right-click over the cross section grid, and select **Material Thickness** from the shortcut menu.
4. In the **Material Thickness** dialog box, enter a thickness value in **Identify thickness for selected part**.

Note:

If the part already has a thickness value assigned, the value appears as the default value.

5. (Optional) Click **Apply to all** to apply the thickness value to all sectioned parts.

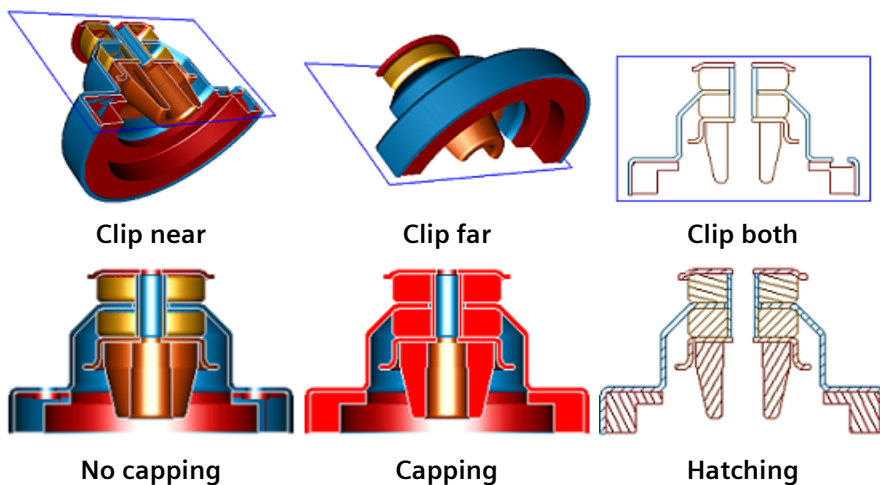
Tip:

If **Apply to all** is selected, the specified value will automatically be assigned to all remaining parts.

Overview of clipping options

You can clip the geometry on either side of the plane to reveal hidden geometry. You can change the appearance of clipped geometry in several ways:

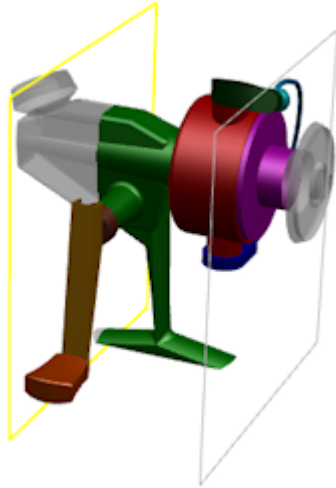
- Apply clipping styles to clipped areas by setting capping and hatching cross section preferences.
- Add end caps to clipped areas to prevent the model from looking hollow or to identify areas where parts overlap.
- Add hatched lines to clipped areas (with or without capping enabled) to differentiate one part from another.



The clip state is saved for each Viewing window and applied to the active plane in that view. When you select a cross section plane, the current view's clip state will be applied to it.

Clipping limitations

- **Display modes** —
 - All of the sections in a view share the same display mode, as determined by the **Show Clipped Geometry** preference on the **Clipping** tab of the **Cross Section Preferences** dialog box. You cannot have different display types for different sections in the same view.
 - If you apply a display mode to clipped geometry, performance may be affected in the Viewing window.



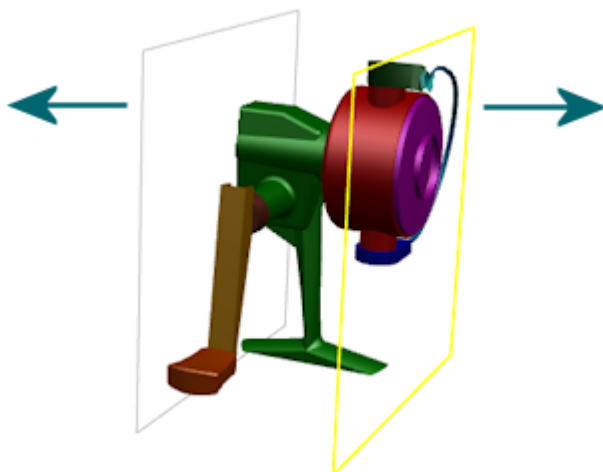
Sections with the Shaded display mode applied

- **Overlapping clipping regions —**

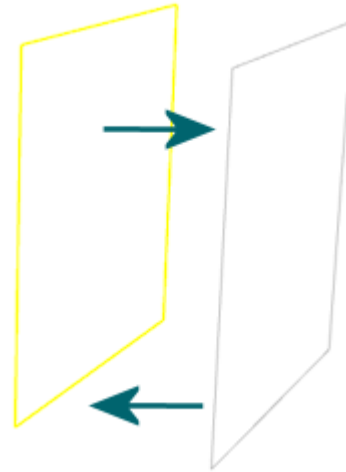
The behavior for sections with overlapping clipping regions can be unexpected. This is true for both sections that are applied to an entire model and sections applied to selected parts.

Example:

Two parallel clipping planes that face away from each other work as expected. But if the planes face *toward* each other, the clipping regions completely intersect and all the geometry in the model is clipped. This means that you cannot clip a slice out of the middle of a model (using cross sections). To do that, create a clipping volume.



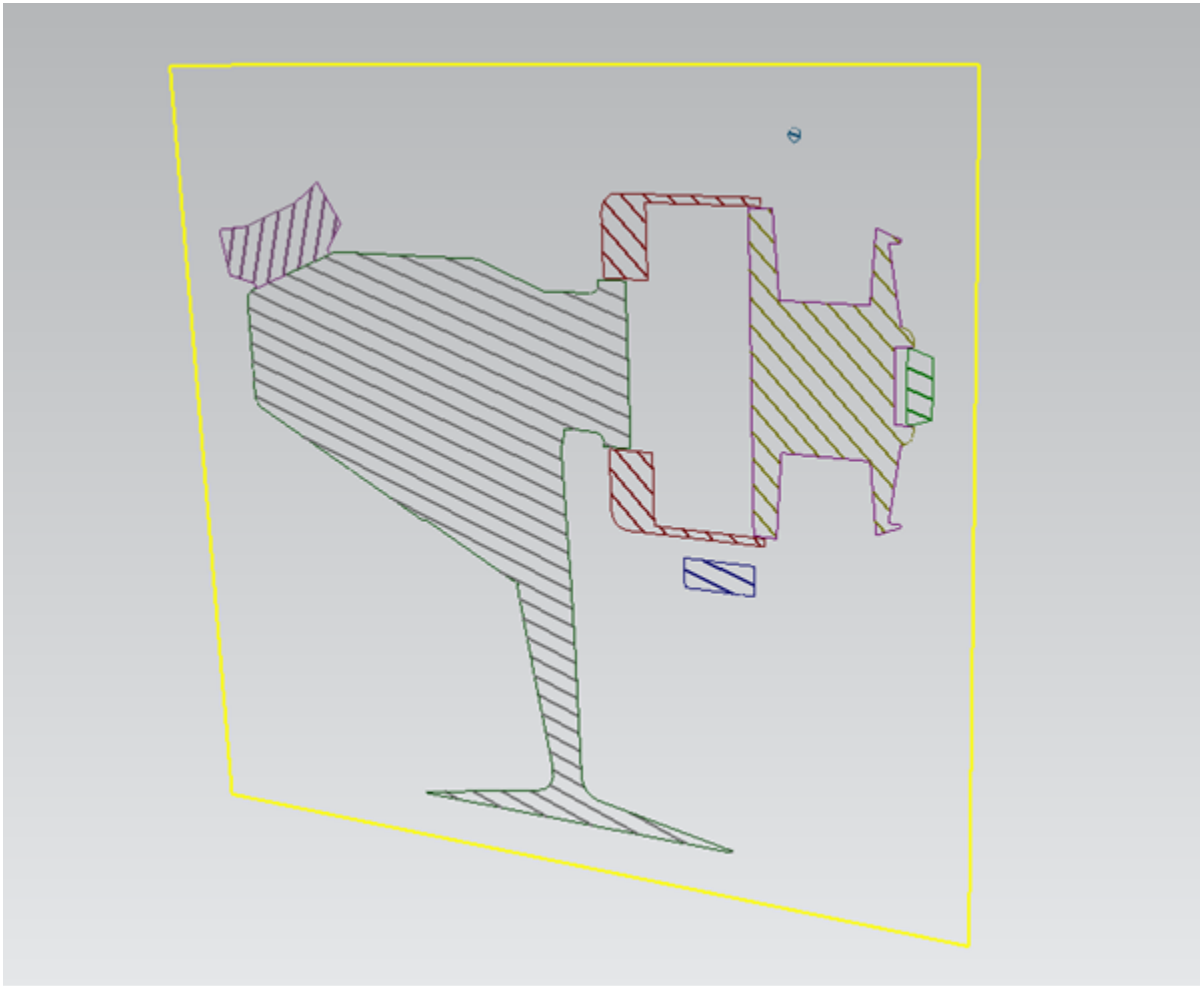
Clipping planes not facing



Clipping planes facing

- **Clip Both —**

When a cross section is in **Clip Both** mode, the hatching can be displayed but not the capping.



- **Number of clipping planes** —

A maximum of 6 clipping planes per view can be displayed at any one time.

Clip cross sections

Clip cross sections to reveal hidden geometry.

Tip:

- Use cross section preferences to draw end caps and to specify display modes for clipped geometry.
- You can display the section plane without borders by choosing **Section3D→Hide Borders**.

- When you choose **Clip Both** to clip geometry from both sides of the cross section, section lines are shown if **Fast Mode** is not activated, even if the **Draw Section Lines** preference is not activated. If **Fast Mode** is activated, click **Manual Update**  on the **3D Section** toolbar to show section lines.

- Create and position the cross section plane.
- (Optional) Set the cross section preferences for clipping.
- Do one of the following:
 - On the **3D Section** toolbar, click a clipping option.
 - In the Viewing window, right-click a cross section, point to **Clipping**, and then select a clipping option.

To	Choose this option
Clip geometry on the near side of the section (with respect to the current view)	Clip Near
Clip geometry on the far side of the section (with respect to the current view)	Clip Far
Clip geometry from both sides of the cross section, leaving only the section plane	Clip Both

- (Optional) To turn off clipping, choose active clip option again.

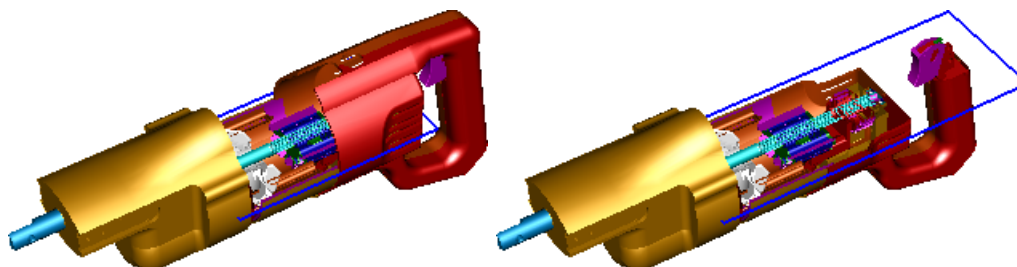
Add or remove parts from a cross section

After you create a cross section, you can add or remove parts and update the section.

- Do one of the following:

To	Do this
Add or remove parts from a cross section	- Do one of the following: - Right-click the section plane and choose Select Parts. - Select the section plane (if you have more than one) and choose Section3D→Select Parts. The sectioned parts are selected.

To	Do this
	2. Hold the Ctrl key as you click parts to add or remove. 3. Do one of the following: • Right-click the section plane and choose Update Parts. • Choose Section3D→Update Parts. • On the Cross Sections page of the Project Workspace window, right-click the plane and select Update Parts.
Select parts to be in the updated cross section	1. (If you have more than one cross section) select the section plane to update. 2. Select the parts to be sectioned. 3. Do one of the following: • Right-click the section and choose Update Parts. • Choose Section3D→Update Parts. • On the Cross Sections page of the Project Workspace window, right-click the plane and select Update Parts.



Using the Cross Section viewer

Overview of using the Cross Section Viewer

You can display the active section or section series in the Cross Section Viewer, a 2D window that is separate from the 3D Viewing window. Use the Cross Section Viewer to **visualize** and **measure** section edges and grid lines.

Display sections in the Cross Section Viewer

1. Create a **section** or **section series**.
2. On the **3D Section** toolbar, click **Show in Viewer** .

The Cross Section Viewer appears, displaying the section edges.

Tip:

Choose **Display Active Section in Viewer** on the Viewer page of the **Cross Section Preferences** dialog box to display the Cross Section Viewer automatically each time a section or section series is created.

3. Use commands in the Cross Section Viewer and shortcut menu to manipulate the view in the following ways:

To	In the viewer, click	Right-click and choose
Fit the section lines in the Viewer.		Fit All
Zoom to a specified area.		Zoom Area
Move the image horizontally or vertically.		Pan
Use the mouse to zoom in or out of the image.		Zoom
Delete any measurements displayed in the Viewer.		Clear All Measurements
Specify cross section preferences.		Preferences
Rotate the image counter clockwise.		N/A
Rotate the image clockwise.		N/A
Flip the image horizontally.		N/A
Flip the image vertically.		N/A
Create a new 2D image of the section information.	N/A	Send to 2D

To	In the viewer, click	Right-click and choose
Create a new 3D document of the section information.	N/A	Send to 3D
Specify properties for the current Cross Section Viewer.	N/A	Properties

4. Manipulate the position of the section plane in the following ways:

To	Do this
Move the section plane along the specified axis.	Click the position slider and move it left or right
Move the section plane along the specified axis.	Enter a coordinate value, and click Apply .
Change the axis.	Select X , Y , or Z .

Tip:

Choose **Always Update the Active 3D View When Slider Moves** in the **cross section preferences** to dynamically update the location of the cross section plane in the 3D Viewing window when you adjust the position slider at the bottom of the Cross Section Viewer.

Perform measurements in the Cross Section Viewer

- On the **3D Measurement** toolbar, click **3D Measurement** .
- Use standard **3D measurement tools** to measure the cross section.

Work with cross sections in the Cross Sections view

You can use the **Cross Sections** view to manipulate sections in the active 3D Viewing window.

When you create a cross section or section series a text entry appears on the Cross Sections page. For axis-aligned sections or sections of a section series, entries are named according to their position on the axis to which they are aligned. For sections along a curve, entries are named according to their offset value.

Use the **Cross Sections** view to manipulate sections in the following ways:

To	Do this
Select a section or section series	Click the name.
Control which sections are displayed	Click the check box.

Delete a section or section series	Right-click a name, and select Delete .
Create a 2D image from a section or section series	Right-click a name, and select Send to 2D .
Create a 3D document from a section or section series	Right-click a name, and select Send to 3D .
Update a section after adding or removing parts	Right-click a name, and select Update Parts .
Adjust section properties	Right-click a name, and select Properties .

Working with PMI cross sections

PMI cross sections appear in the **PMI view** along with other **PMI**. Some PMI cross sections, including those created in NX 7.5 or later as Lightweight Sections, are associated with specific parts.

You can do the following with PMI cross sections:

- Turn them on and off in the PMI view.
- Change their appearance **properties** (Professional and Mockup).
- Turn **clipping** on and off. PMI cross sections that are associated with parts, only clip their associated parts.
- Select or clear the **Show dynamic section border** check box in the **PMI Preferences** dialog box to show or hide section lines and section borders.
- Create a live 3D section from a PMI section using the context menu in either the Model Views or PMI view.

You cannot do the following with PMI cross sections:

- Delete them.
- Reposition them.
- Realign them.
- Add or remove parts from them.

Note:

Cross-section hatching is displayed the same way as it is displayed in NX. The PMI data must come from NX 9.0 or later.

Measure cross sections

Note:

- When you measure section edges in 3D models with **NURBS data**, picked edges are highlighted. When you measure section edges in 3D models without NURBS data, approximate points are highlighted when you attempt to pick an edge.
- In Professional or Mockup, to display section edges, you must select **Draw Section Lines** in the cross section preferences.

Tip:

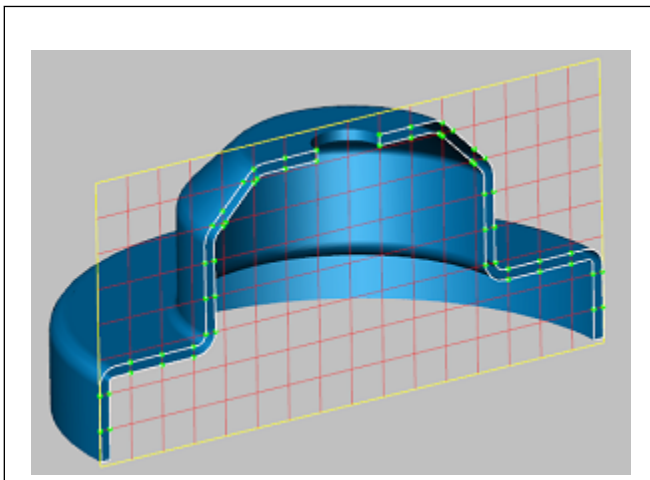
When measuring cross sections, **clip** geometry from both sides of the section plane (from one side of the section plane in Base and Standard).

1. Create and position a section plane.
2. On the **3D Measurement** toolbar, click **3D Measurement** .
3. Use **3D measurement tools** to measure the cross section.

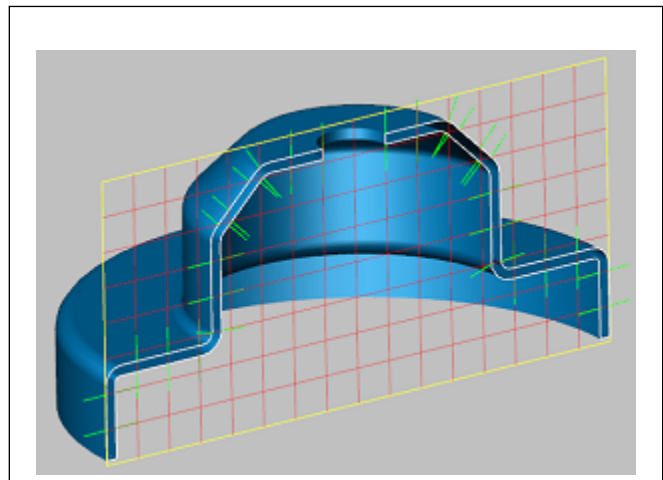
Creating point sheets

Overview of creating point sheets

Cross section points indicate where section lines intersect with grid lines. Section point sheets can be either graphical representations (points or normals) or a point sheet report listing the points of contact. You can **create a section point sheet** for a section or section series that is orthogonal to the X, Y, or Z axis.

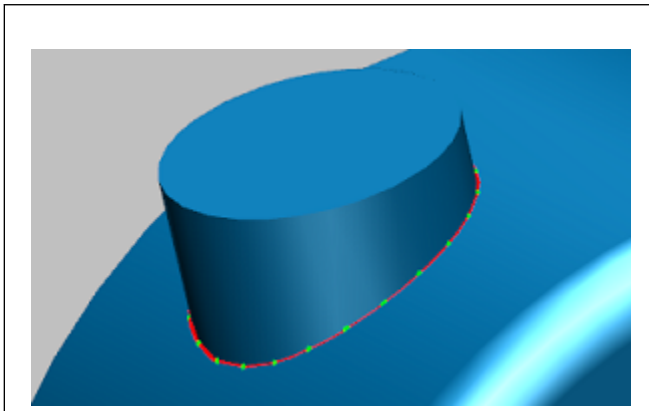


Section point sheet points



Section point sheet normals

Curve points are points generated along single or multiple connected curves. You can **create a curve point sheet** to add points and generate a point sheet report.



Curve point sheet points

X	Y	Z	I	J	K
90.500000	15.000000	23.048861	0.000000	0.000000	0.000000
92.408890	14.734710	23.219362	0.000000	0.000000	0.000000
94.094162	14.006785	23.665564	0.000000	0.000000	0.000000
95.480606	12.918639	24.276718	0.000000	0.000000	0.000000
96.578598	11.471402	24.993137	0.000000	0.000000	0.000000
97.280533	9.739087	25.717701	0.000000	0.000000	0.000000
97.498451	7.852718	26.354973	0.000000	0.000000	0.000000
97.189621	5.938657	26.851089	0.000000	0.000000	0.000000
96.364044	4.177323	27.180876	0.000000	0.000000	0.000000
95.120827	2.741895	27.362959	0.000000	0.000000	0.000000
93.498985	1.674958	27.448946	0.000000	0.000000	0.000000
91.503096	1.085973	27.478556	0.000000	0.000000	0.000000

X Offset = 0.000000 ; Y Offset = 0.000000 ; Z Offset = 0.000000

Point sheet report

Create point sheets for a section or section series

1. Select a section or section series that is orthogonal to the X, Y, or Z axis in the 3D scene.
2. If the section lines are not displayed, do the following:
 - a. On the **3D Section** toolbar, click **Manual Updated** .
 - b. If section lines are still not displayed, right-click the section, choose **Properties**, and on the **General** tab of the **Cross Sections Properties** dialog box, select the **Draw Section Lines** check box.
3. Choose **Section3D→Create Point Sheets→Section**.

The **Section Point Sheet** dialog box appears, and the **X Axis**, **Y Axis**, or **Z Axis** check boxes are selected depending on the orientation of the section plane. For example, if the section plane is parallel to the XY plane, then **X Axis** and **Y Axis** are selected.

4. (Optional) To change how the intersection points are generated, select **X Axis**, **Y Axis**, or **Z Axis**.

For example, if you select **X Axis**, and the section plane is parallel to the XY or XZ plane, intersection points will be generated where the section lines intersect X axis grid lines.

5. (Optional) Enter offset values in **X**, **Y**, or **Z**, from the world origin, for use in generating the point report.

Note:

These offsets will not be used when previewing the points in the 3D scene.

6. To display the intersection points as dots, click **Show Points**.

Note:

You can also display the sections and points in the secondary window, but curves and curve points do not appear.

7. To display short line segments at each intersection showing the direction of the surface normal at the point, click **View Normals**.
8. To generate a point report, click **Report**.

The **Report Definition Wizard** appears, in which you can preview and print the report.

Create point sheets for single or multiple connected curves

1. Choose **Section3D→Create Point Sheets→Curve**.
2. In the **Select Curve(s)** dialog box, select input curves.
3. (Optional) To deselect the last selected curve, click **Reset Last**.
4. (Optional) To deselect all of the selected curves, click **Reset All**.
5. Click **Done**.
6. In the **Curve Point Sheet** dialog box, type the **Number of Points** to generate along the curve.
7. (Optional) Type offset values in **X**, **Y**, or **Z**, from the world origin, for use in generating the point report.

Note:

These offsets will not be used when previewing the points in the 3D view.

8. Click **View Points** to see the points in the Viewing window.
9. Click **Report**.

The **Report Definition Wizard** appears, in which you can preview and print the report.

Capture cross sections as 3D documents

You can capture cross section edges and send them to a new 3D document.

When you capture the section edges for a single cross section, the section appears as a single part in the new 3D document. When you capture the edges for a section series, the new 3D document has an assembly with a part for each section in the series.

Caution:

Captured cross section data is not **precise**, so for models with NURBS or B-Rep data, measuring the captured cross section will not be as accurate as measuring the original 3D section.

1. Create one or more cross **sections** or section **series**.

2. Select a section or section series.

3. Do any of the following:

- Choose **Section3D→Send to 3D**.

If you have more than one cross section, the **Send to 3D** dialog box appears, and you can choose to create a 3D document from only the active cross section or from all cross sections in the Viewing window.

- In the 3D Viewing window, right-click the section plane and choose **Send to 3D**.
- From the Cross Section Viewer, right-click and choose **Send to 3D**.
- In the **Cross Sections** view, right-click the section name and choose **Send to 3D**.

A new 3D document opens with the cross section information represented as wireframe data.

If the section includes grid lines, grid labels, and a title, they also appear in the new 3D document.

Capture cross sections as 2D images

You can capture cross section edges and send them to a new 2D image.

When you capture the section edges for a single cross section, the edges are on a markup layer in the new 2D image. When you capture the edges for a section series, the new 2D image has a markup layer for each section in the series.

1. Create one or more cross **sections** or section **series**.

2. Select the section or section series to capture.

3. Do any of the following:

- Choose **Section3D→Send to 2D**.
- In the 3D Viewing window, right-click the section plane and choose **Send to 2D**.
- From the Cross Section Viewer, right-click and choose **Send to 2D**.
- In the **Cross Sections** view, right-click the section name and choose **Send to 2D**.

The model edges that intersect with the cross section plane appear as a new 2D vector image.

If the section includes grid lines, grid labels, and a title, they also appear in the new 2D image.

Set cross section preferences

Set preferences for new cross sections. Settings are preserved across work sessions.

1. Do one of the following:

- On the **3D Section** toolbar, click **Preferences** .
- In the Cross Section Viewer, right-click and select **Preferences**.

The **Cross Section Preferences** dialog box appears.

2. On the **General** tab, change any of these options:

In this section	Use this option	To do this
Section Lines	Draw Section Lines	Display section lines when you clip geometry from the sides of the section plane.
	Draw Lines on Top	Display section lines on top of all model parts so that lines are never hidden.
	Color	Select the color for section lines.
	Width	Specify the width of section lines.
Section Border	Color for Active Section	Choose a color for the active cross section border.
	Width	Choose the width for the active section border.
	Color for Other Sections	Choose a color for the borders of sections that are not selected.
	Width	Choose the width for the borders of sections that are not selected.

In this section	Use this option	To do this
Slider and Manipulator Control	User Defined Step Size	Specify the distance the active cross section plane moves each time you press the arrow keys or drag the Position Plane slider. By default, slider steps are 1/100 (0.01) of the distance normal to the plane in the visible scene. Note: For sections along curves, any user defined step value is ignored. The value on the slider ranges from 0 (start of curve) to 1 (end of curve). The slider step size is always 0.01 (1/100 of the distance normal to the plane in the visible scene).
	Automatically Show Slider	Show the slider when the cross section tools are active.
	Automatically Show Manipulator	Show the manipulator when the cross section tools are active.
Performance	Display Section Lines for Moving Planes (faster when off)	Specify whether or not to render section lines while moving section planes. Section lines and any clip capping update when you stop moving the slider. When sectioning a large model, turn this off to enhance performance.
	Fast Mode (requires manual update for line, caps and hatching)	Specify whether or not to delay the rendering of section lines, capping, and hatching until you choose Manual Update . Turn this on to improve performance when creating or moving section planes in very large models.
	Draw dashed border when update is needed	Use dashed lines for section borders when you are in Fast Mode and cross section are not fully rendered.
6-DOF Input Device	Use World Coordinate System	Base on the orientation of the object on the X-, Y-, and Z-axes. Note: This option works only for 3DConnexion devices, not for other immersive tracking devices.
	Use Screen Coordinate System	Base on the orientation of the object with respect to the computer screen. Note: This option works only for 3DConnexion devices, not for other immersive tracking devices.
Point Sheet Points	Color	Select the color of the displayed points.
	Width	Select the width of the displayed points.

3. On the **Clipping** tab, change any of these options:

In this section	Use this option	To do this
When Clipping is Off	Show Plane	Show or hide the section plane.
	When Clipping Both	Display section lines using the part color.
Capping	Automatically Show Plane View	Automatically display clipped sections in the plane view.
	Enable Capping	Prevent clipped parts of the model from looking hollow by including end caps. Note: Enabling capping requires more system resources and may slow performance in the Viewing window.
	Cap Solids Only	Apply capping only to geometric solids.
	Capping Color	Specify the end cap color and determine if it will be Per Part or Global .
	Intersecting Regions	Specify the end cap color for the area within sections where parts intersect.
Hatching	Enable Hatching	Add hatch lines to capped parts.
	Spacing	Enter a value to determine the spacing for hatch lines. Select Multiply by Part Volume to apply the spacing of hatch lines based on each part's bounding box volume.
	Line Color	Specify the line color and determine if it will be Per Part or Global .
Show Clipped Geometry As	Shaded	Show clipped geometry as shaded.
	Transparency	Show shaded clipped geometry as semi-transparent. You can specify the degree and color of the transparency in the 3D viewing Preferences dialog box. Note: Enabling transparency requires more system resources and may slow performance in the Viewing window.
	Construction Geometry	Show wireframe construction geometry.
	Tessellation Lines	Show clipped geometry as tessellated. You also can specify the color of the lines.
	Hidden Lines	Hidden lines are tessellation lines on areas of the model that are behind the tessellation lines in the foreground.

In this section	Use this option	To do this
		You can specify the following viewing options for hidden lines: Invisible: Makes hidden lines invisible. Ghosted: Makes hidden lines faint in appearance. Visible: Displays hidden lines.

4. On the **Grid** tab, change any of these options:

In this section	Use this option	To do this
Not applicable	Show Grid	Display cross section grid lines for axis aligned sections.
Lines	Grid Spacing	Choose the spacing between grid lines. These units are the same as those used in the model file.
	Width	Select the width of the grid lines.
	Color	Select the grid line color.
	Transparency	Move the slider to choose the transparency of the grid display.
Labels	Show Grid Labels	Display numeric labels on the grid.
	Show Title	Display a section title consisting of the section number, alignment axis, and location along the axis.
	Size	Choose the text size of the grid labels and titles. Note: Grid labels cannot overlap. If you choose a size that is too large, the text adjusts so the labels do not run together.
	Color	Specify the color of the grid labels and titles.
	Refit during navigation	Have labels and titles update to remain in the view when you zoom and pan while viewing the section in Plane View. Depending on the view, labels and titles appear either on the borders of the grid, or the edges of the Viewing window.

5. On the **Viewer** tab, change any of these options:

In this section	Use this option	To do this
Not applicable	Display Active Section in Viewer	Automatically display the Cross Section Viewer when you create a section or section series.
Cross Section Viewer	Background Color	Choose the background color of the Cross Section Viewer.
	Always Update the Active 3D View When Slider Moves	Move the section plane in the 3D Viewing window when you use the slider in the Cross Section Viewer.

6. On the **Material Thickness** tab, change any of these options:

In this section	Use this option	To do this
Not applicable	Enable Material Thickness	Enable functionality to display a virtual line simulating material thickness for any new cross sections. Tip: When you enable material thickness through the Cross Section Properties dialog box, it is only enabled for the section you selected.
	Line Style	Select the line type to create for the offset line. The offset line will be drawn in the same color and width as the section line.
	Label offset lines	Creates annotation labels on the active cross section plane that displays the thickness value of the created offset lines.
	Offset in face normal direction	Creates an offset line on the same side as the surface normal.
Action to take for parts that have no thickness attribute	Action	Select from one of the following options if a node has missing thickness attribute information. Use Default value: Applies the value listed in Default value to parts with missing thickness attributes. None: If the part does not have a thickness attribute, the thickness value is set at zero and not displayed. Prompt: A dialog box appears for any parts that do not have thickness attributes. Supply a thickness value for the part. Note: If you enter a large value for Prompt (as compared to the model size), the visual result might be compromised. For example, if your surface model is 10x10mm and you enter a thickness value of 1000, it will be

In this section	Use this option	To do this
		difficult to find the thickness offset line visually on screen.
	Default value	Enter a value that applies to missing thickness attributes. Note: This value only applies when you select Use Default value from Action . You can also change material thickness values for specific parts.

Change the appearance of a cross section

- Do one of the following:
 - In the 3D Viewing window, right-click a section, and choose **Properties**.
 - In the Cross Section Viewer, right-click and choose **Properties**.
 - In the **Cross Sections** view, right-click a cross section, and choose **Properties**.

The **Cross Section Properties** dialog box appears.

- On the **General** tab, change any of these options:

In this section	Use this option	To do this
Section Lines	Draw Section Lines	Display section lines when you clip geometry from the sides of the section plane.
	Draw Lines on Top	Display section lines on top of all model parts so that lines are never hidden.
	Color	Select the color for section lines.
	Width	Specify the width of section lines.

- On the **Clipping** tab, change any of these options:

In this section	Use this option	To do this
When Clipping is Off	Show Plane	Show or hide the section plane.
When Clipping Both	Show Lines in Part Color	Display section lines using the part color.
	Automatically Show Plane View	Automatically display clipped sections in the plane view.

4. On the **Grid** tab, change any of these options:

In this section	Use this option	To do this
Not applicable	Show Grid	Display cross section grid lines for axis aligned sections.
Lines	Grid Spacing	Choose the spacing between grid lines. These units are the same as those used in the model file.
	Width	Select the width of the grid lines.
	Color	Select the grid line color.
	Transparency	Move the slider to choose the transparency of the grid display.
Labels	Show Grid Labels	Display numeric labels on the grid.
	Show Title	Display a section title consisting of the section number, alignment axis, and location along the axis.
	Size	Choose the text size of the grid labels and titles. Note: Grid labels cannot overlap. If you choose a size that is too large, the text adjusts so the labels do not run together.
	Color	Specify the color of the grid labels and titles.
	Refit during navigation	Have labels and titles update to remain in the view when you zoom and pan while viewing the section in Plane View. Depending on the view, labels and titles appear either on the borders of the grid, or the edges of the Viewing window.

5. On the **Material Thickness** tab, change any of these options:

In this section	Use this option	To do this
Not applicable	Enable Material Thickness	Enable functionality to display a virtual line simulating material thickness for any new cross sections.

In this section	Use this option	To do this
		Tip: When you enable material thickness through the Cross Section Properties dialog box, it is only enabled for the section you selected.
	Line Style	Select the line type to create for the offset line. The offset line will be drawn in the same color and width as the section line.
	Label offset lines	Creates annotation labels on the active cross section plane that displays the thickness value of the created offset lines.

Creating cross section reports

Overview of creating cross section reports

Generate cross section reports for single sections, sections along curves, and a series of cross sections. Cross section reports can include analysis of the section area or perimeter, moment of inertia, center of gravity, and other calculated cross section properties.

Note:

Only **Perimeter** and **Area** are available if the model does not contain XT B-Rep data.

Create section reports

1. Create a cross section or a series of cross sections.
2. Choose **Section3D→Report→Preferences**.
3. In the **Reference Axes** section of the **Report Preferences** dialog box, select one of the following:

Choose this option	To do this
Use reference axes in previous report generation when applicable	When you begin to generate the report, if the current section normals are identical to the section normals from the last report that was run, automatically use the reference axes from the last report; otherwise, create a reference coordinate system at the section centroid with the option to reposition it.
Define or use a coordinate system via Coordinate System feature	When you begin to generate the report, define a new reference coordinate system or select an existing coordinate system via the Manage Coordinate Systems dialog box.

4. Click **OK** to close the **Report Preferences** dialog box.
5. On the **Section3D** toolbar, click **Create Report** .
6. In the **Report Contents** dialog box, select the check boxes for the information that you want to include in your report.

Note:

In the **General** section, select one or more of these options:

- **Perimeter**

The measurement of the perimeter of the cross section.

- **Effective Area**

The area of the total cross section after certain portions of the cross section with different modulus of elasticity than those of the rest of the cross section have been adjusted to account for the mixed moduli. The adjustment is made by multiplying the value of the portion's dimension that is parallel to the axis of interest by the ratio of its own modulus of elasticity to the reference modulus of elasticity. This applies when the cross section contains different materials. One of the materials is designated as the reference material, having the reference modulus. The effective area is based on the reference material modulus.

- **Area**

The area of the total cross section without any adjustments to the dimensions of any portion of the cross section.

- **Centroid**

The point at which the area of the section is evenly distributed.

7. Click **Next**.
8. (This step only applies if your report includes **Principle Axes**, **Moments of Inertia**, or **Principal and Reference Axes**.) Depending on your selection in the **Reference Axes** section in the **Report Preferences** dialog box, do one of the following:

If you chose this option	Do this
Use reference axes in previous report generation when applicable	Do one of the following: • If the section normals in this report are the same as the normals from the last report that was run, the reference axis setting from the last report is automatically applied. Continue to the next step.

	If the section normals in this report do not match the normals from the last report, a reference coordinate system appears at the section centroid. Reposition it as needed, and then in the Create Report – Reference Axis dialog box, click Next.
Define or use a coordinate system via Coordinate System feature	- Choose Actions→Coordinate Systems→Manage. - In the Manage Coordinate Systems dialog box, define a new coordinate system or select an existing one. - Click Toggle Working CS to make it the Working Coordinate system, and then click OK. - In the Create Report – Reference Axis dialog box, click Next.

- In the **Report Definition Wizard**, view the preview of the report data, and then click **Next** to continue or **Cancel** to cancel the report.
- In the **Create Report Output** dialog box, set your report format options, specify the location for the report output file, and click **Finish**.

Set cross section report preferences

Set cross section report preferences to specify

- Reference axes
- Young's modulus
- Color of principle axes
- Unit adjustment

- Choose **Section3D→Report→Preferences**.
- In the **Report Preferences** dialog box, change the preferences in one of these ways:

Note:
Some of the options may be disabled if the model has no B-Rep data.

In this section	Use this option	To do this
Reference Axes	Use reference axes in previous	Apply the reference axis setting from the previous report, when the section normals in the two reports are identical.

In this section	Use this option	To do this
	report generation when applicable	
	Define or use a coordinate system via Coordinate System feature	When prompted, define a new references axis or select an existing coordinate system via the Manage Coordinate Systems dialog box.
Young's Modulus	Source Attribute	Set the attribute name for the Young's modulus value.
	Force Units	Force units and model units (read from the JT file) construct the unit of Young's modulus.
	Action for parts missing Young's modulus attribute	Select from one of the following options for a part in the section that has missing Young's modulus attribute information. • None: No action will be performed and the report generation stops. • Use default value: Applies the value listed in Default Value to parts with missing Young's Modulus attribute information. • Prompt: The Material Young's Modulus dialog box appears for any parts that have missing Young's modulus attributes. Click Apply to all for the specified value to be assigned to all remaining parts without further prompting.
	Default value	Enter a value that applies to parts that are missing Young's modulus attribute information. Note: This value only applies when you select Use default value from Action for parts missing Young's modulus attribute.
	Reference	Choose one of the following for Reference Young's Modulus: • Use default value: The value from Default Value is used. • Retrieve from selected part: During report generation, you will be prompted to select a part to use as the Young's modulus reference.

In this section	Use this option	To do this
Color of Principal Axes	X Axis, Y Axis, and Z Axis	Select the colors of the principle axes to appear in the 3D scene.
	Allow to adjust units before report is saved	A dialog box appears before the report is saved to notify you to adjust the units. Adjust units in the 3D Measurement Preferences dialog box on the Calculations page.

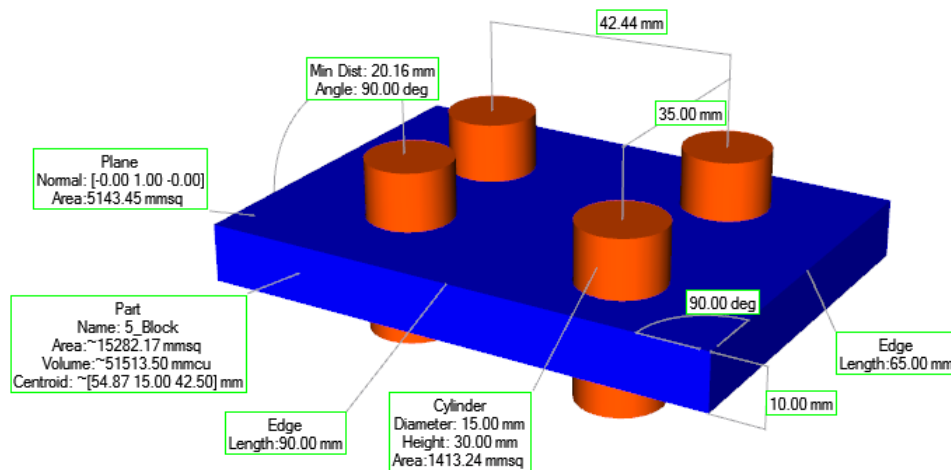
3. Click **OK**.

Your changes are applied to section reports.

Measuring parts

Overview of measuring parts

You can measure your 3D models in various ways.



Different measurement types display different measurement information.

- **Single measurement** displays information about a single part feature.
- **Double measurement** displays the distance or angle between two part features.
- **Chain measurement** displays a series of double measurements from one part feature to the next.
- **Fan measurement** displays a series of double measurements from the first part feature to each subsequent feature.

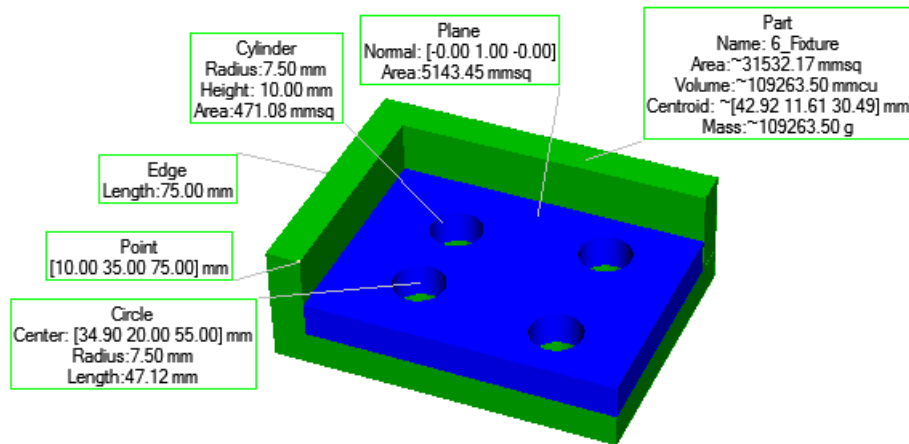
- **Distance measurement** displays the distance between two part features.
- **Angle measurement** displays the angle between two part features.
- **Radius measurement** displays the radius of a circular edge or surface.
- **Local radius measurement** displays the radius of a point on an edge or a point on a surface.
- **Diameter** measurement displays the diameter of a circular edge or surface.
- **Part summation measurement** displays the total area, volume, and mass (optional) of a selected set of parts.
- **Bounding box measurement** displays the bounding box (with its measurements) of a selected set of parts.
- **Part set measurement** displays the minimum or maximum distance between two sets of parts.

Note:

- Most measurement types require you to select specific **part features**.
- For most measurement types, you can see step-by-step instructions on the Status bar.

Measuring single part features

Overview of measuring single part features



There are several ways to measure single part features:

- **Radius** measurements display the radius of a circular edge or surface.

- **Diameter** measurements display the diameter of a circular edge or surface.
- **Single** measurements display information for any part feature. The information displayed depends on the feature type.

Feature	Pick	Information displayed
Point, Vertex, Arc Center, Midpoint, or Point on Edge	All	Point coordinates (Optional) Angle measurements to XY, ZY, and XZ planes
Edge	All Straight line Circle Circular arc Spline curve	Type of edge Length, (optional) midpoint, (optional) intersection point Center, radius or diameter, length Center, radius or diameter, angle, length Length Note: You can set a preference to display either radius or diameter for circles and circular arcs.
Surface	All Plane Cone Cylinder Sphere Surface of revolution	Type of surface, area Normal, area Height, radius or diameter, area, angle Height, radius or diameter, area Radius or diameter, area, center coordinates Radius 1, radius 2, height, area Note: • You can set a preference to display either radius or diameter for cones, cylinders, and spheres. • Cylinder height is determined by the distance between the two farthest points.
Part	All	Name of part Surface area Volume

Feature	Pick	Information displayed
		Centroid (single part only) Mass (Optional)

Tip:

You can display the total length for multiple edges, the total area for multiple faces, and the total area, volume, and mass (optional) for multiple parts. Hold Ctrl as you select the edges, faces, or parts with measurements you want to add.

Measure single part features

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. (Optional) Change **measurement preferences**.
3. On the **3D Measurement** toolbar, click **Single Measurement** .
4. Choose a part feature to measure in one of these ways:
 - Select the **part feature** to measure.
 - **Calculate** a point.

The part feature you select is highlighted and information about the feature is displayed in a label near the feature.

The information displayed is relative to the **Working Coordinate System**.

5. (Optional) To add multiple single (edge, surface, or part) measurements, hold Ctrl as you select the part features. The features must all be of the same type. Multiple picks of the same part feature are ignored. The measurement label displays the sum of the measurements, indicated by <sum> and the number of part features.

Note:

- For a single part measurement, volume and mass are not displayed if the part has no volume or mass data.
- You can see step-by-step instructions on the Status bar.

Measure radius or diameter

You can display the radius or diameter of circular edges or cylinders.

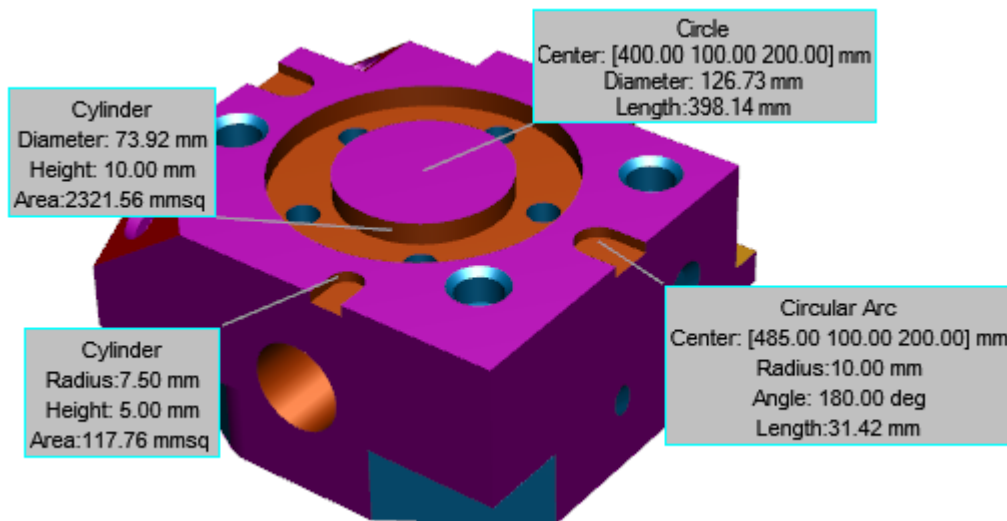
1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. (Optional) Change **measurement preferences**.
3. Do one of the following:

To display the	Do this
Radius	On the 3D Measurement toolbar, click Radius  .
Diameter	On the 3D Measurement toolbar, click Diameter  .

4. Select the **circular edge or cylinder** to measure.

The information is displayed in a label near the feature.

In Standard, Professional, and Mockup, the location displayed is relative to the **Working Coordinate System**.



Note:

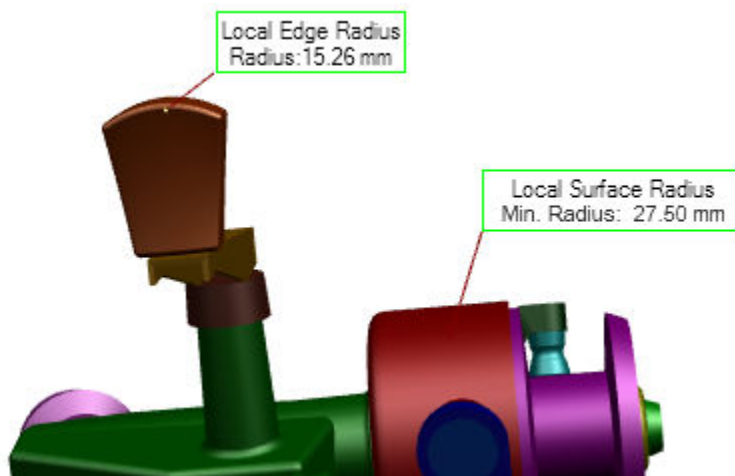
You can see step-by-step instructions on the Status bar.

Measure local radius of points on edges and surfaces

You can display the local radius of points on edges and points on surfaces. When you measure a point on an edge, the measurement is the local radius at the point. When you measure a point on a surface, the measurement is the minimum radius at the point. If you measure a point on a straight edge or a flat surface, the measurement is displayed as **INF** (infinite radius).

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. (Optional) Change **measurement preferences**.
3. On the **3D Measurement** toolbar, click **Local Radius** .
4. Select the **point on an edge or the point on a surface** to measure.

The information is displayed in a label near the feature.

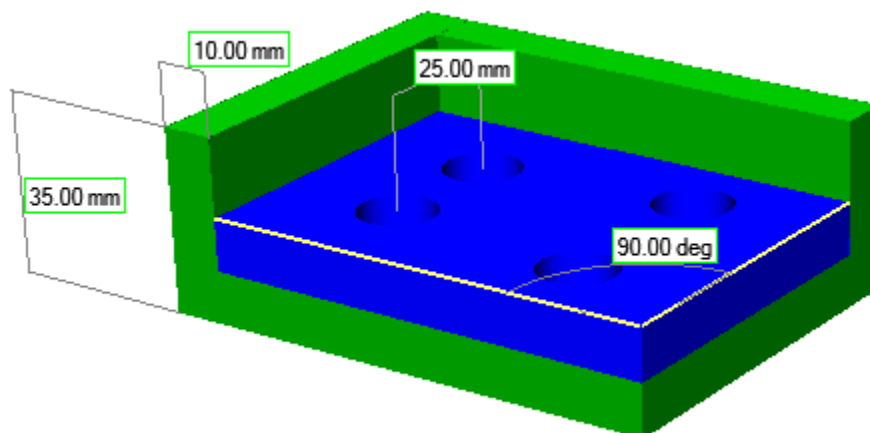


Note:

You can see step-by-step instructions on the Status bar.

Measuring distances and angles between part features

Overview of measuring distances and angles between part features



Measurements between Edge and Surface features behave as follows:

- Measurements to a circular edge are made to the center of the circle.
- Measurements to a straight edge (line) are made to the infinite line containing the edge.
- Measurements to a planar face are made to the infinite plane containing the face.
- Measurements to a cylindrical or conical face are made to the infinite line containing the axis of the cylinder or cone.
- Measurements to a spherical face are made to the center of the spherical surface.
- Double measurements of two straight edges or cylinder axes display minimum distance if they are parallel, and angle if they intersect.
- Double measurements of two straight edges or cylinder axes that are not parallel and do not intersect (not coplanar) display both minimum distance and angle.

Measure the distance between part features

There are two ways to measure distances:

- **Distance** measurement displays only distances. You can measure between two parts or part features.
- **Double** measurement displays the distances and angles depending on the types of features you select.

1. On the **3D Measurement** toolbar, click **3D Measurement** .

2. Do one of the following:

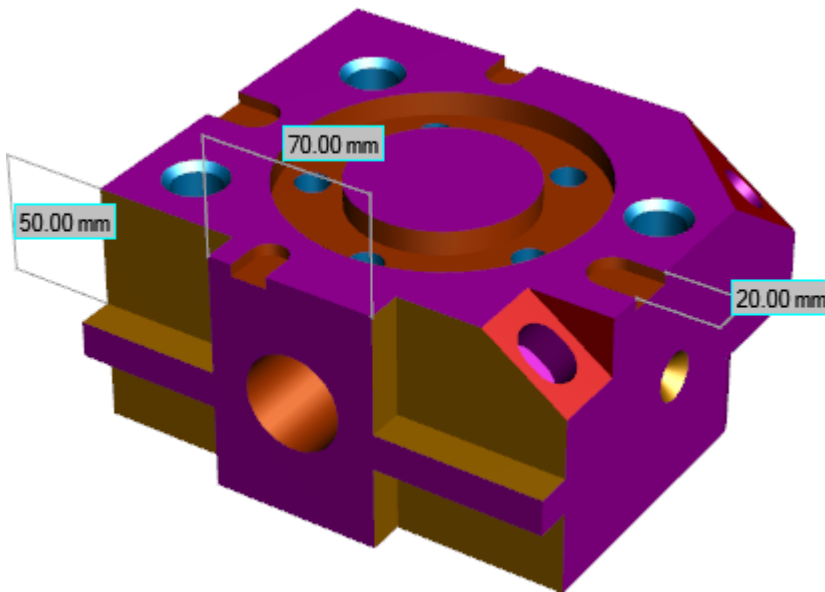
To use	Do this
Distance measurement	On the 3D Measurement toolbar, click Distance  .
Double measurement	On the 3D Measurement toolbar, click Double  .

3. In Standard, Professional, and Mockup, choose a part feature to measure in one of these ways:

- Select the first **part feature**.
- **Calculate** a point.

4. Select the second part feature on your model. You can specify a different feature type.

A measurement label appears between the part features. You can drag to move the label.



Note:

You can see step-by-step instructions on the Status bar.

Measure the angle between part features

There are two ways to measure angles:

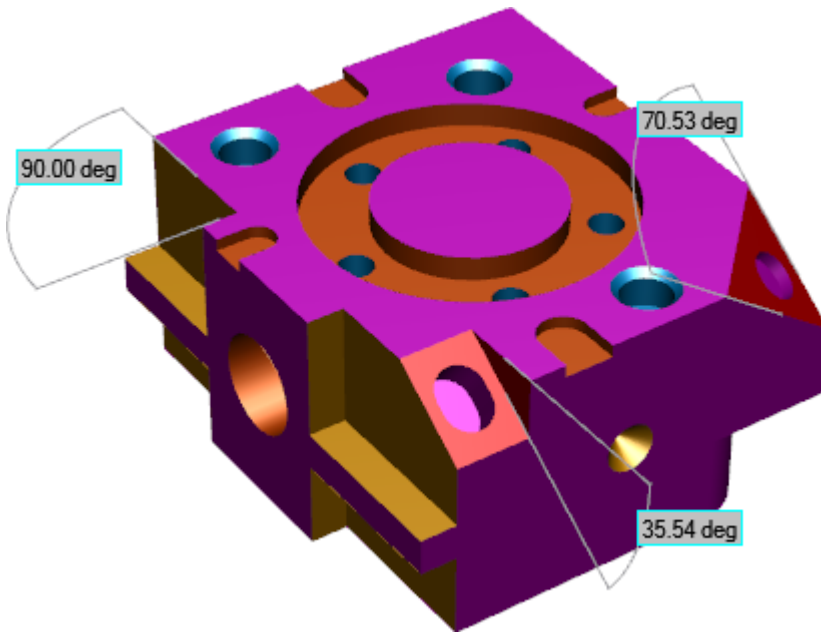
- **Angle** measurement displays only angles.
- **Double** measurement displays angles and distances depending on the types of features you select.

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. Do one of the following:

To use	Do this
Angle measurement	On the 3D Measurement toolbar, click Angle  .
Double measurement	On the 3D Measurement toolbar, click Double  .

3. Select the first **edge or surface**.
4. Select the second edge or surface. You can specify a different feature type.

A measurement label appears between the part features. You can drag to move the label.



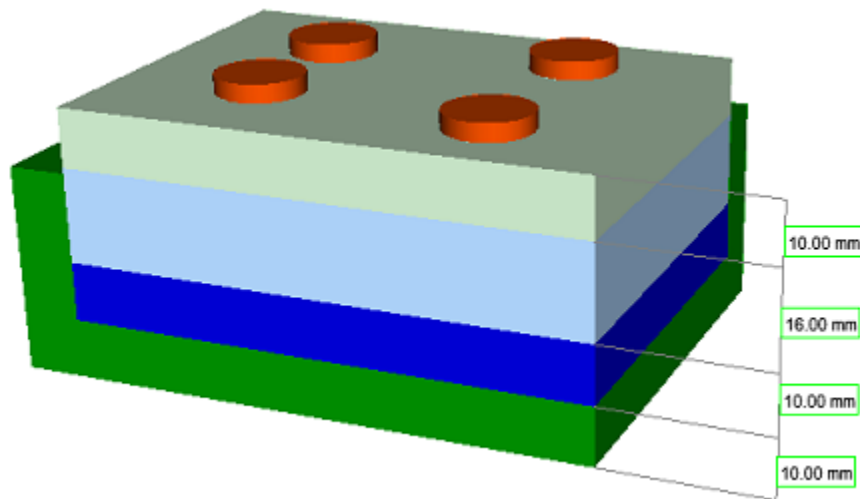
Note:

You can see step-by-step instructions on the Status bar.

Measuring between multiple features (Chain)

Overview of measuring between multiple features (Chain)

Chain measurements are a type of double measurement that measures between every part feature you select and the previous feature. In this example, the first feature picked was the bottom corner vertex.



Measure between multiple features (Chain)

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Chain And Fan** , and then choose **Chain**.
3. Specify the first part feature in one of these ways:
 - Select the **part feature**.
 - **Calculate** a point.

4. Select the second part feature. You can specify a different feature type.

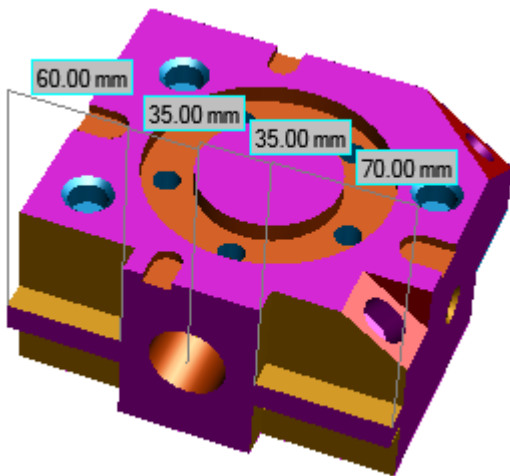
A measurement label appears between the first and second part features.

5. Select a third feature.

A measurement label appears between the second and third part feature

6. (Optional) Continue to select part features.

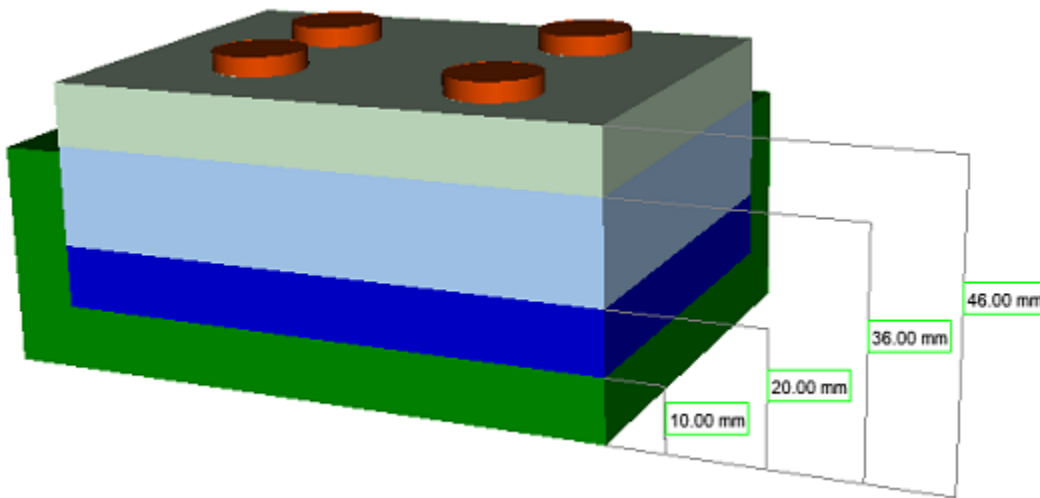
With each click, a measurement label appears between that feature and the previous feature.



Measuring from one feature to multiple features (Fan)

Overview of measuring from one feature to multiple features (Fan)

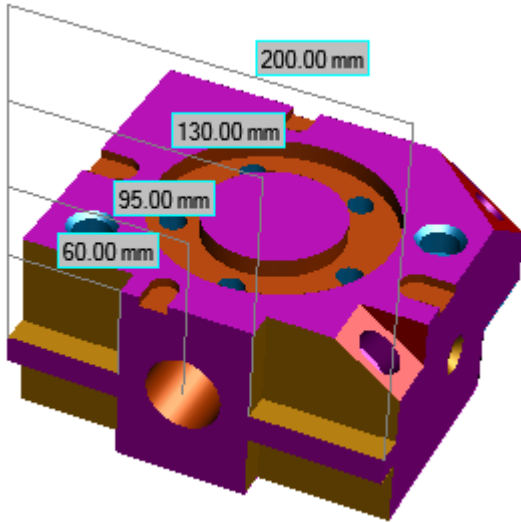
Fan measurements are a type of double measurement that measures between the first part feature you select and every subsequent feature. In this example, the first feature picked was the bottom corner vertex.



Measure from one feature to multiple features (Fan)

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Chain And Fan** , and then choose **Fan**.
3. Specify the first part feature in one of these ways:
 - Select the **part feature**.
 - **Calculate** a point.
4. Select the second part feature. You can specify a different feature type.
A measurement label appears between the first and second part features.
5. Select a third feature.
A measurement label appears between the first and third part features.
6. (Optional) Continue to select part features.

With each click, a measurement label appears between that feature and the first feature.



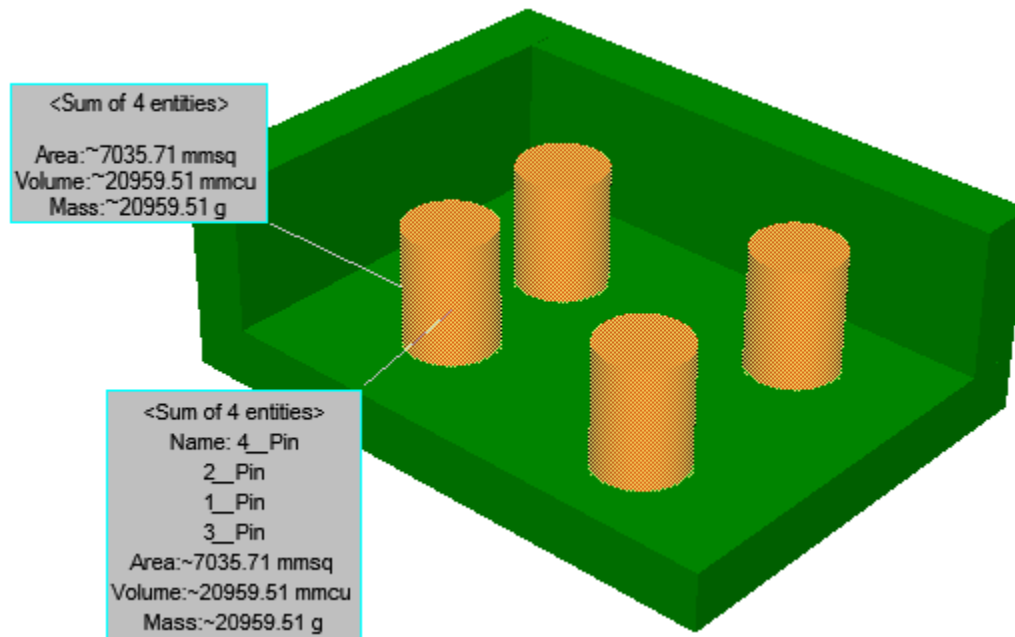
Measuring area, volume, and mass for a set of parts

Overview of measuring area, volume, and mass for a set of parts

You can select a set of parts and display their total area, volume and mass (optional). You can set a measurement preference to include part names in the measurement label and to limit the number of part names to display. You can set a measurement preference to include the mass.

Note:

Volume and mass totals are not displayed if one or more parts have no volume or mass data.



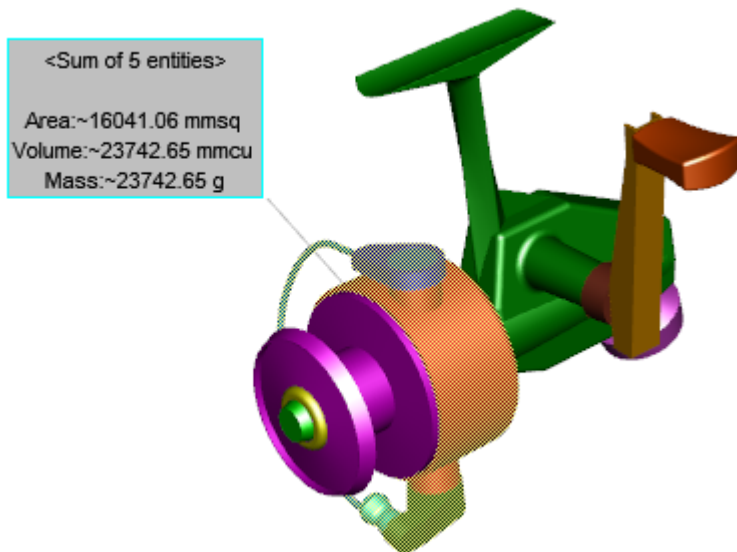
Measure area, volume, and mass for a set of parts

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Preferences** .
3. In the **Measurement Preferences** dialog box on the Calculations page, do the following in the **Mass and Density** section:
 - a. Select the **Display mass in Part Measurements** check box.
 - b. If the model attributes from the CAD package include the mass, for **Source Attribute**, select the attribute.
 - c. For **Mass Units**, choose the default unit of measure to apply to the density setting for parts that lack attributes with the necessary mass information.
 - d. For **Default Density**, type a default value to calculate the mass for parts that lack attributes with the necessary mass information.
4. (Optional) Set your measurement preferences to display the list of part names in the measurement label.
5. On the **3D Measurement** toolbar, click **Part Summation Measurement** .

The **Select Parts** dialog box appears.

6. **Select** the parts in the Viewing window or the assembly tree. You can hold Ctrl as you select multiple parts.
7. In the **Select Parts** dialog box, click **Finish**.

The measurement label with the total area, volume, and mass appears next to the last part. You can drag to move the label.



Note:

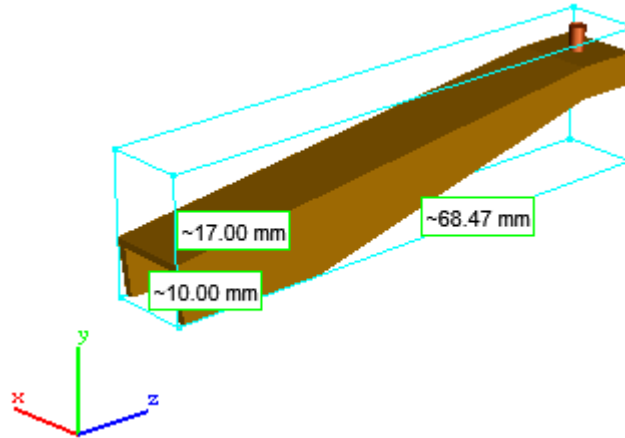
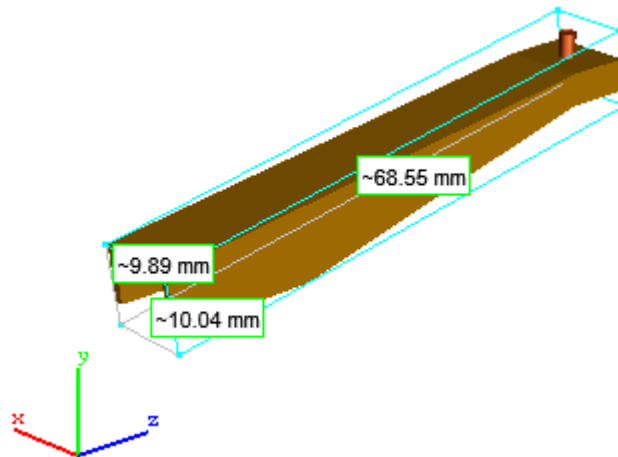
Volume and mass totals are not displayed if one or more parts have no volume or mass data.

Displaying the bounding box for selected parts

Overview of displaying the bounding box for selected parts

You can select one or more parts and display their bounding boxes and their measurements. You can **set a measurement preference** to display either of the following types of bounding boxes:

- **Axis Aligned:** Displays the smallest bounding box that is aligned to the X, Y, Z axes global coordinate system.
- **Approximate Best Fit:** Displays a compact bounding box that encloses the parts. However, the most compact bounding box possible is not guaranteed. Additionally, the measurement values can change based on the orientation of the 3D model.

Axis Aligned bounding box**Approximate Best Fit** bounding box**Display the bounding box for selected parts**

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. (Optional) Set your **measurement preferences** to calculate either **Axis Aligned** or **Approximate Best Fit** bounding box.

Note:

For a **Approximate Best Fit** bounding box, the measurement values can change based on the orientation of the 3D model.

3. On the **3D Measurement** toolbar, click **Bounding Box Measurement** .

The **Select Parts** dialog box appears.

4. **Select** one or more parts to include in the bounding box. To select multiple parts, hold Ctrl as you select the parts.

5. In the **Select Parts** dialog box, click **Finish**.

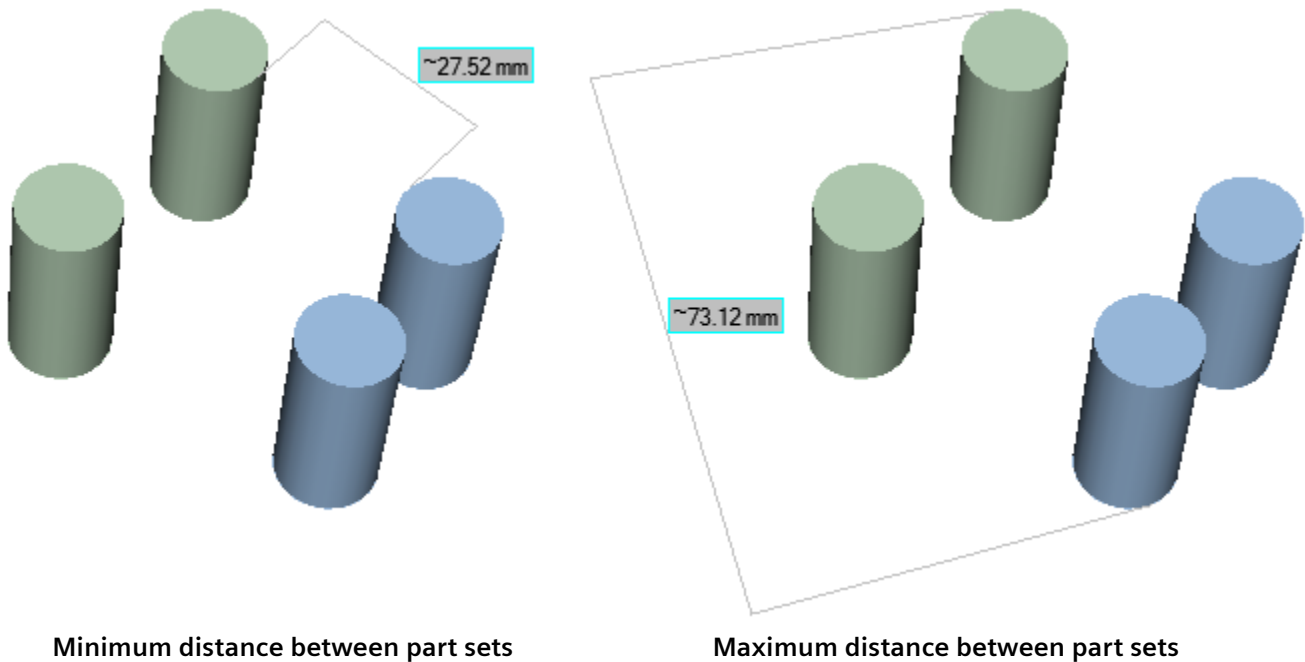
The bounding box appears with its three measurement labels. You can drag to move the labels.

Measuring between sets of parts

Overview of measuring between sets of parts

You can specify two sets of parts (or one set of parts and a part feature) and measure the minimum or maximum distance between them.

- A minimum distance part set measurement displays the distance between the two closest parts (one in each set).
- A maximum distance part set measurement displays the distance between the two furthest parts (one in each set).
- Part set measurements are not associative measurements.



Measure between two sets of parts

Specify the two sets of parts and measure the minimum or maximum distance between them.

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Part Set Measurement** .

The **Select Parts** dialog box appears.

- Do the following to specify a minimum or maximum distance measurement:

To	Do this
Display the minimum distance measurement	On the 3D Measurement toolbar, click Compute Minimum Distance  .
Display the maximum distance measurement	On the 3D Measurement toolbar, click Compute Maximum Distance  .

Note:

The default is to display the minimum distance between part sets.

- Select** one or more parts to include in the first part set. To select multiple parts, hold Ctrl as you select the parts.
- In the dialog box, click **Next**.
- Confirm that the **Part Set** option is active.
- Select one or more parts to include in the second part set.
- Click **Finish**.

The measurement result is displayed.

Measure between a set of parts and a feature

Specify the set of parts and a part feature, and measure the minimum or maximum distance between them.

- On the **3D Measurement** toolbar, click **3D Measurement** .
- On the **3D Measurement** toolbar, click **Part Set Measurement** .

The **Select Parts** dialog box appears.

- Do the following to specify a minimum or maximum distance measurement:

To	Do this
Display the minimum distance measurement	On the 3D Measurement toolbar, click Compute Minimum Distance  .
Display the maximum distance measurement	On the 3D Measurement toolbar, click Compute Maximum Distance  .

Note:

The default is to display the minimum distance between part sets.

4. **Select** one or more parts to include in the first part set. To select multiple parts, press Ctrl as you select the parts.
5. In the dialog box, click **Next**.
6. In the dialog box, click **Individual Entity**.
7. Specify the entity **type**, and then click the entity.
8. Click **Finish**.

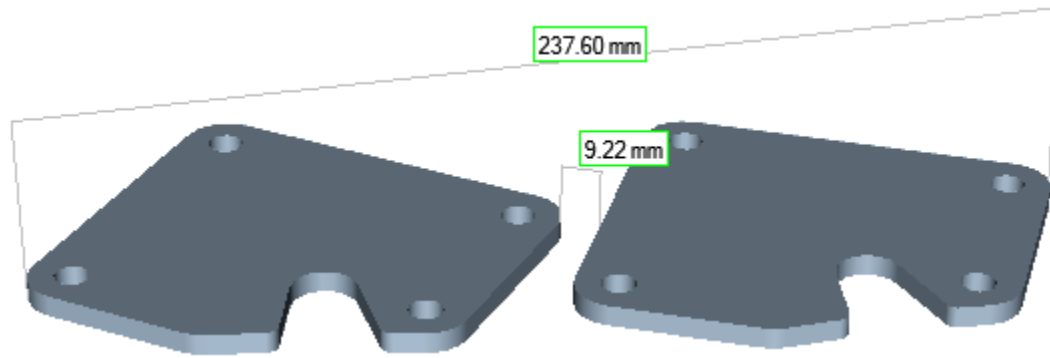
The measurement result is displayed.

Measuring minimum and maximum distances

Overview of measuring minimum and maximum distances

Distance and **part set** measurements can display minimum or maximum distances. A minimum distance measurement displays the distance between the two closest points. A maximum distance measurement displays the distance between the two furthest points.

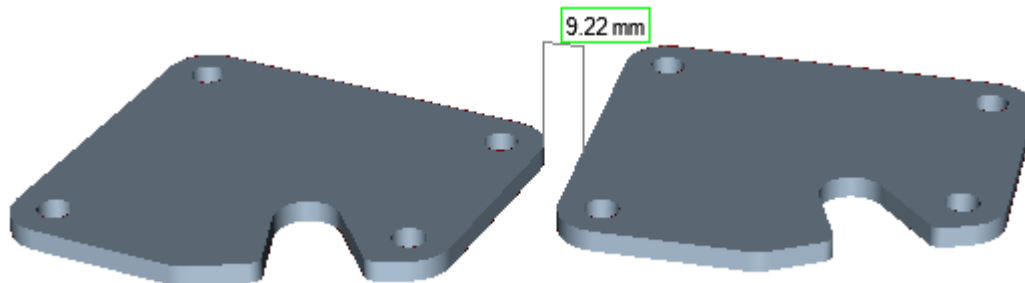
For Distance or Double measurements, at least one part feature should be an edge, surface or part. When measuring between point features, the minimum and maximum distances are the same.

**Note:**

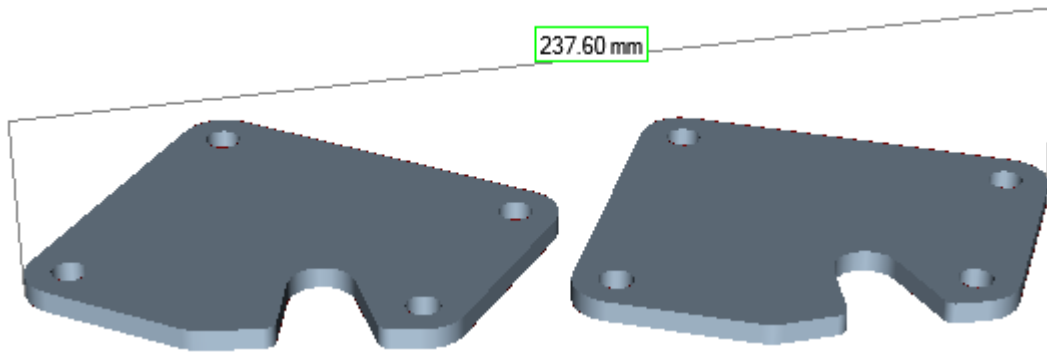
When measuring to a circular edge, measurements show the minimum or maximum distance between the feature and the edge curve instead of the center of the curve.

Measure minimum distances

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Minimum Distance** .
3. Measure the distance between **two part features** or **part sets**.

**Measure maximum distances**

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Maximum Distance** .
3. Measure the distance between **two part features** or **part sets**.



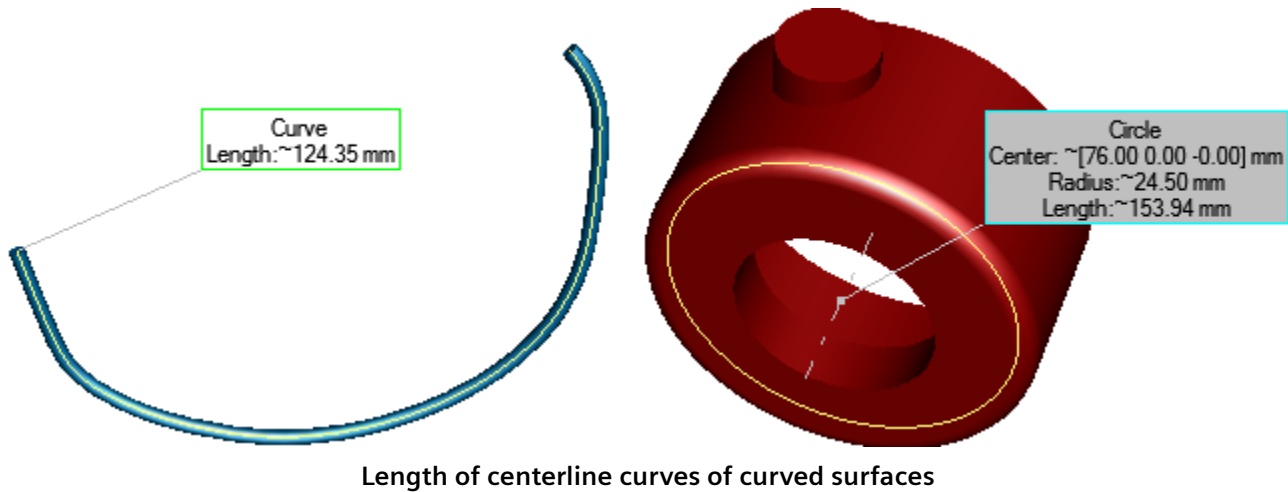
Measuring centerline curves of surfaces

Overview of measuring centerline curves of surfaces

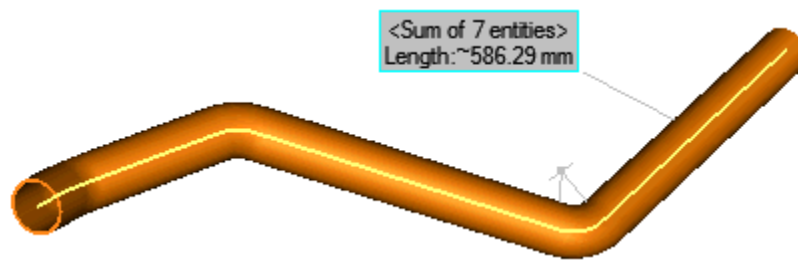
You can measure the centerline curve of a single curved surface or calculate the cumulative length of curves of multiple surfaces. The centerline curve of a surface is the curve along which another curve was swept to generate the surface. Centerline curves can only be computed for those surfaces that are generated by sweeping a circular arc along another curve.

Note:

- To perform centerline curve measurements, your model must contain JT B-Rep or XT B-Rep data.
- The computed centerline curve may not exactly match the directrix curve used to generate the surface. Therefore, the result of centerline measurement is only approximate even though NURBS data is used for computing it.



Length of centerline curves of curved surfaces



Sum of several centerline curves

Measure centerline curves of surfaces

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **Measurement** toolbar, click **Single Measurement** .
3. On the **3D Selection** toolbar, hold Ctrl and click **Surface**  to make it the only active selection type.
4. Choose **Measurement**→**Measure Surface Centerline**.
5. Click a surface.

The measurement of the surface length along its centerline appears.

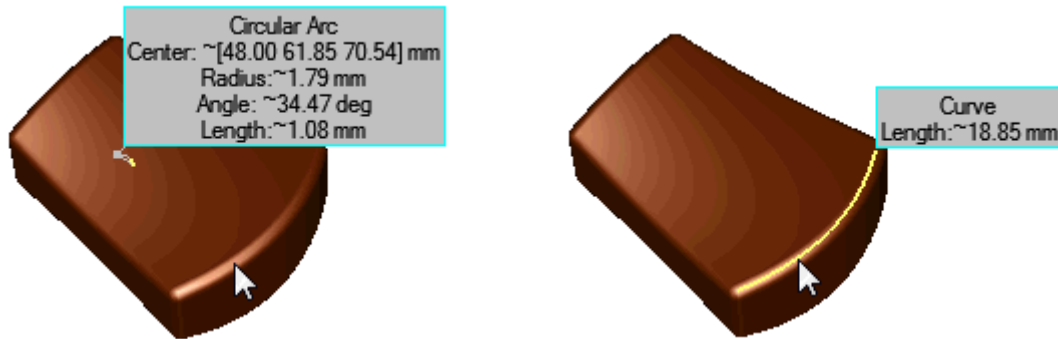
Tip:

To measure the cumulative length of the curves of multiple surfaces, hold Ctrl as you select the surfaces.

Note:

- Some surfaces have two centerline curves. In those cases, if the first curve measured is not the curve you want, hold Ctrl and select the surface again to measure the second curve.

Curve one and curve two of same surface



- When surfaces are converted from JT B-Rep data to XT B-Rep data a single surface can sometimes be split into two or more smaller surfaces. To get the complete centerline curve for cases like this, you may have to pick the surface multiple times.

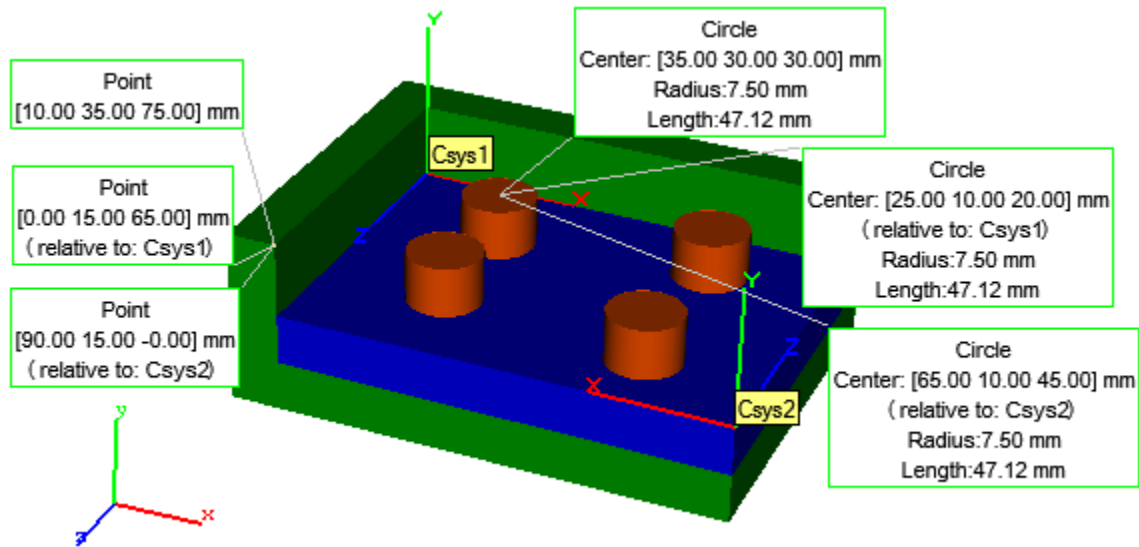
Measuring relative to a coordinate system

Overview of measuring relative to a coordinate system

Several types of measurements are relative to the coordinate system. When you open a model, you are working in a Global coordinate system. You can define new coordinate systems and select one to be the Working Coordinate System (Working CS). The Working CS is used for measurements. If you do not select a Working CS, measurements are relative to the Global coordinate system.

The following types of measurements are relative to either the Global coordinate system or the Working CS:

- Single measurements of points, vertices, circular edges, planar surfaces, non-cylindrical surfaces, and parts.
- Distance measurements when displaying the X, Y, Z delta values.
- Angle measurements when displaying extrapolated intersection points.



Choose a coordinate system for measurements

You can choose any PMI or User Defined coordinate system to be the Working Coordinate System (Working CS) and use it for measurements. If you do not designate a Working Coordinate System, the Global coordinates system is used for measurements.

1. (Optional) Create a **coordinate system**.
2. Do one of the following to choose the Working CS:

To	Do this
Choose a visible coordinate system	In the Viewing window, right-click a coordinate system and choose Toggle Working CS .
Choose from a list of coordinate systems	a. On the 3D Coordinate System toolbar, click Manage Coordinate Systems . b. In the Manage Coordinate Systems dialog box, select the name of the coordinate system to use and click Toggle Working CS. c. Click OK.

3. Perform a **single**, **distance**, **chain**, or **fan** measurement.

Note:

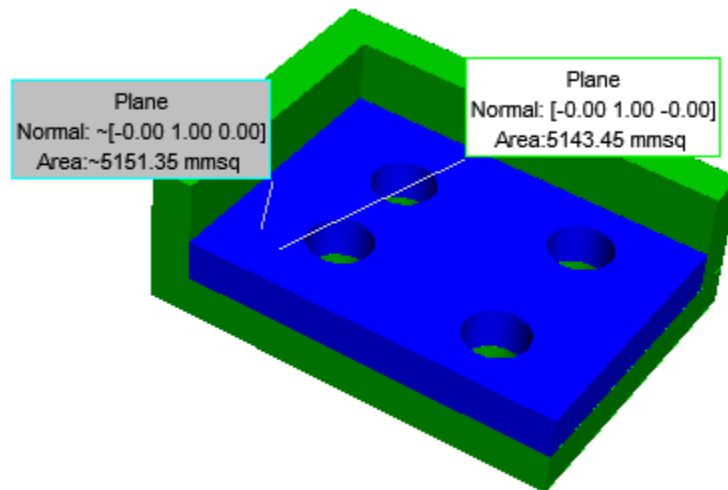
The coordinate system remains the working coordinate system until you specify a new working coordinate system.

4. (Optional) Choose another coordinate system to be the Working CS and perform more measurements.
5. (Optional) To return to using the Global coordinate system, do one of the following:
 - In the Viewing window, right-click the Working CS and choose **Toggle Working CS**.
 - Open the **Manage Coordinate Systems** dialog box, select the name of the Working CS, and click **Toggle Working CS**.

Displaying approximate or exact measurements

Overview of displaying approximate or exact measurements

If your model is created with Boundary Representation (B-Rep) data, you can display either exact measurements or approximate measurements. The default is to measure the precise geometry and display exact measurements, but you can choose to measure the tessellated data and display approximate measurements.



Note:

- Only models that include precise geometry (B-Rep data) can be measured with exact measurements.
- Interactivity may be slower when you use exact measurements.
- Approximate measurements are indicated by a tilde (~) in the measurement display label.
- Semi-precise measurements that use Ultra-Lightweight Precise (ULP) data are indicated by a tilde and an equals sign (~=) in the measurement display label.

Circle Center: [475.00, 97.00, 150.00] mm Diameter: 20.00 mm Length: 62.83 mm	Circle Center: ~[475.00, 97.00, 150.00] mm Diameter: ~20.00 mm Length: ~62.83 mm	Circle Center: ~[475.00, 97.00, 150.00] mm Diameter: ~20.00 mm Length: ~62.83 mm
Exact	Semi-precise	Approximate

Tip:

To determine whether your model includes precise geometry, right-click a part. If the options **Show Vertices** and **Show Edges** are available, the model includes precise geometry (B-Rep data). If the options are unavailable, the model does not include precise geometry.

Display approximate or exact measurements

Choose to display approximate or exact measurements when measuring models containing precise geometry (B-Rep data).

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. Choose **Measurement** → **Use Approximate** to disable precise measurements, or clear **Use Approximate** to use precise measurements.

Tip:

- To determine whether your model includes precise geometry, right-click a part. If the options **Show Vertices** and **Show Edges** are available, the model includes precise geometry (B-Rep data). If the options are unavailable, the model does not include precise geometry.
- Setting **Use Approximate** when doing associative measurements will result in measurements that do not move or update when objects are moved.

Manipulating measurement display labels

Select and move a measurement label

1. In the Viewing window, click a measurement label.

The measured part feature is highlighted.

2. Drag the label anywhere in the Viewing window.

Note:

You can move only one measurement label at a time.

Display the supplemental angle

1. Click the measurement label of an angle.
2. Drag the measurement label to the area of the supplement of the angle that is currently measured.

The measurement changes to measure the supplemental angle.

Delete all 3D measurements

On the **3D Measurement** toolbar, click **Clear** .

Note:

In Base, measurement labels that appear as a result of opening a session file are not deleted when you choose **Clear**.

Delete individual 3D measurements

Right-click a measurement label, and choose **Delete**.

The selected measurement is deleted, but other measurements remain unchanged.

Note:

- In Mockup, if **Display Active Measurement** is active and you delete the active measurement, the data for the most recently created double measurement is displayed.
- You can use the Undo command to undo the last measurement Delete action, but not the last Clear action.

Undo your last part feature pick

As you pick features for measurements, you can undo the last pick.

On the **3D Measurement** toolbar, click **Undo Pick** .

The last feature you picked is deselected. If the pick completed a measurement, the measurement is cleared.

Copy measurement label text

Right-click the display label, and select **Copy Text**.

The text is copied to the clipboard. You can paste the text into another place within the program that accepts text, or any Windows application that accepts pasted text.

Set measurement preferences

Set the measurement preferences to specify the

- Units of measurement and precision with which measurements are calculated.
- Information displayed in the measurement display labels.
- Ways in which certain part features are measured.
- Appearance of measurement display labels. (Standard, Professional, and Mockup, only.)
- Treatment of measurements that are stored in session files. (Standard, Professional, and Mockup, only.)

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. On the **3D Measurement** toolbar, click **Preferences** .
3. In the **Measurement Preferences** dialog box, change the measurement preferences in one of these ways:

On this tab	Use this option	To do this
Display	Precision	Specify the number of decimal places to display in measurements. The measurement is calculated to the number of decimal places you specify.
	Only show the last _ measurements	Specify the maximum number of display labels. Select the check box, and then type a value.
	Display interpolated midpoints Standard, Professional, and Mockup, only.	Include the coordinates of the midpoint when measuring distances.
	Display the X,Y,Z delta values	Include the absolute values of the deltas when measuring distances.

On this tab	Use this option	To do this
	Standard, Professional, and Mockup, only.	If you want to display the values as positive or negative values, where negative values include the minus sign, select the Display them signed check box.
	Display extrapolated intersection points Standard, Professional, and Mockup, only.	On the measurement display labels of Angle measurements of edges that intersect or of edges that would intersect if they were extended, include the coordinates of the intersection point or extrapolated intersection point.
	Display distance (radius) from origin to selected point Standard, Professional, and Mockup, only.	On the measurement display labels of points, vertices, arc centers, midpoints, points on edge, and calculated points, display the distance from the origin of the coordinate system to the point.
	Only show _ Part Names in Single and Part Summation Measurements Standard, Professional, and Mockup, only.	Specify whether to display part names in part measurements and if part names are displayed, specify the maximum number to display.
	Use fixed color scheme for associative measurements Standard, Professional, and Mockup, only.	Specify for all associative measurement labels to have a consistent appearance, with green edges and a white background.
	Display angle to XY plane Standard, Professional, and Mockup, only.	On the measurement display labels of points, vertices, arc centers, midpoints, points on edge, and calculated points, display the angle between the XY plane of the coordinate system and the vector from the origin of the coordinate system to the point or vertex.
	Display angle to YZ plane	On the measurement display labels of points, vertices, arc centers, midpoints, points on

On this tab	Use this option	To do this
	Standard, Professional, and Mockup, only.	edge, and calculated points, display the angle between the YZ plane of the coordinate system and the vector from the origin of the coordinate system to the point or vertex.
	Display angle to XZ plane Standard, Professional, and Mockup, only.	On the measurement display labels of points, vertices, arc centers, midpoints, points on edge, and calculated points, display the angle between the XZ plane of the coordinate system and the vector from the origin of the coordinate system to the point or vertex.
Calculations	Model units	Specify the units of measurement in which the model was created. Warning: Do not change Model Units unless it displays a value of Unknown. Note: • This setting only affects the displayed units of future measurements; it does not affect the units of the underlying document. • Parts inserted into the first loaded model are matched to the document units of the first model by an automatic scaling transformation.
	Display units	Change the units in which measurements are displayed. From the Display Units list, select the units of measurement. Note: If you choose None, measurements are displayed with number values only; no units of measurement are displayed.
	Measure midpoint at	Specify how midpoints are calculated when the Display interpolated midpoints preference is active.

On this tab	Use this option	To do this
	Standard, Professional, and Mockup, only.	
	Measure circle/arc as Standard, Professional, and Mockup, only.	Specify whether to display the radius or diameter for Single measurements of circles and arcs.
	Bounding Box Measurement Standard, Professional, and Mockup, only.	Specify whether to display Axis Aligned or Approximate Best Fit bounding boxes for bounding box measurements. Note: The measurement values can change based on the orientation of the 3D model.
	Mass and Density Standard, Professional, and Mockup, only.	Specify whether to include mass information in measurement results. If you want your part measurements to include mass, you can select the CAD attribute specified in your organization's CAD software to store mass values. For parts without a mass attribute, you can specify default values. - Select the Display mass in Part Measurements check box. - If the model attributes from the CAD package include the mass, for Source Attribute, select the attribute. - For Mass Units, choose the default unit of measure to apply to the density setting for parts that lack attributes with the necessary mass information. - For Default Density, type a default value to calculate the mass for parts that lack attributes with the necessary mass information.

On this tab	Use this option	To do this
Line Standard, Professional, and Mockup, only.	Line style	Specify the style of measurement lines and leader lines.
	Line color	Specify the color of measurement lines and leader lines.
	Thickness	Specify the thickness, in points, of measurement lines and leader lines.
	(Line Ends) Style	Specify whether arrowheads (or other symbols) are added to the ends of lines.
	(Line Ends) Size	Specify the size of symbols added to the ends of lines.
	Display leader lines on top	Always display measurement leader lines on top of models, even when they are behind geometry.
Edge Standard, Professional, and Mockup, only.	Edge style	Specify the style of the borders around measurement labels.
	Edge color	Specify the color of the borders around measurement labels. The default edge color is green for associative measurements.
	Thickness	Specify the thickness, in points, of the borders around measurement labels.
Fill Standard, Professional, and Mockup, only.	Fill style	Specify the style of the fill of measurement labels.
	Fill color	Specify the color of the fill of measurement labels. The default fill color is white for associative measurements.
	Transparency	Specify the level of transparency (from Transparent to Opaque) of the fill of measurement labels.
Font	Font	Specify the font used in measurement labels.
	Font Style	Specify the style of the font used in measurement labels.
	Size	Specify the size of the font used in measurement labels.
	Underline or Strikeout	Specify special font effects used in measurement labels.
	Color	Specify the color of the font used in measurement labels.

On this tab	Use this option	To do this					
Highlight Standard, Professional, and Mockup, only.	Highlight Color	Specify the selection color of the features you measure.					
	Thickness	Specify the thickness, in pixels, of the selection highlight on the features you measure. You can specify a thickness value of 1-10.					
General Standard, Professional, and Mockup, only.	Save to session files	Specify whether to save measurement data in session files.					
	Restore during session load	Specify whether to display measurements when you open a session file containing 3D measurement data.					
	(Restored measurement) Label	Specify a text string to be added to measurements that have been saved and restored, to indicate that the measurement may be out-of-sync with the 3D data. The text string is added to labels of non-associative measurements and associative measurements that have not been updated.					
	Update during session load or snapshot apply	Control how associative measurements behave when you apply snapshots, open PLM XML files, or open session files. Updating associative measurements requires loading B-Rep data from the JT file, which can be expensive. This preference gives users control over if or when this update occurs. If this preference is <i>not</i> selected, associative measurements will not update after being restored and therefore will always behave like non-associative measurements. The cost of the update will never be incurred. If this preference <i>is</i> selected, the update occurs during the following operations: <table><tr><th>For</th><th>When you</th></tr><tr><td>Snapshots containing associative measurements</td><td>Apply the snapshot.</td></tr><tr><td>PLM XML</td><td>Do any of the following: Open a PLM XML file that was saved with a </td></tr></table>	For	When you	Snapshots containing associative measurements	Apply the snapshot.	PLM XML
For	When you						
Snapshots containing associative measurements	Apply the snapshot.						
PLM XML	Do any of the following: Open a PLM XML file that was saved with a						

On this tab	Use this option	To do this	
		For	When you
			view containing associative measurements.
			• Apply a snapshot containing associative measurements.
		Session files	Do any of the following: • Select Measurement. • Apply a snapshot containing associative measurements. • Apply a 3D geometry asset containing associative measurements.
	(Measurement Reports) Save to session files	Specify whether to save measurement reports when you save a session file.	
	Reset to Default	Change all preferences on the General tab to the default settings.	

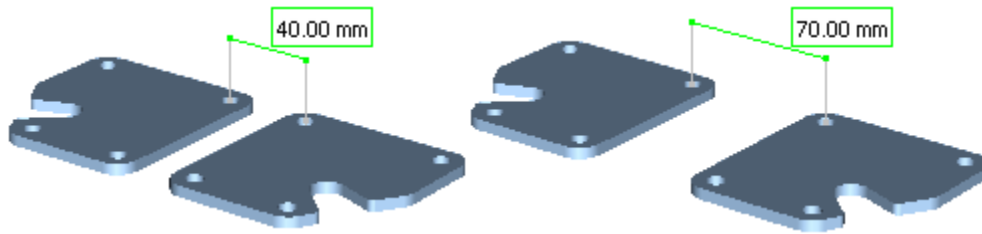
- Click **OK**.

New measurement display labels reflect your preferences. Existing measurement display labels do not change.

Overview of associative measurements

Associative measurements are 3D measurements that remember the objects they measure. If the objects move, the measurements move with the objects. If the objects change, the measurements update to reflect the change.

For example, if you measure the distance between two holes in two different parts, each end of the measurement is associated with a hole. If you move one of the parts, the end of the measurement that is associated with that hole moves too. The measurement label updates as the distance changes.



You can only create associative measurements when using data with **B-Rep Entity IDs**. Entity IDs contain a unique number for each face, edge, and vertex of a JT part. Associative measurements use the IDs to remember which part entities they measure. You can use Inspector to **determine if a JT file contains B-Rep Entity IDs**, and therefore supports associative measurements. You cannot create associative measurements on JT data that does not have Entity IDs.

Tip:

Setting **Use Approximate** when doing associative measurements will result in measurements that do not move or update when objects are moved.

Note:

If you have older JT files (JT v9.0 or earlier) with B-Rep Entity IDs, you may need to use **Generate Selection Preview Data** to update the part tessellation to a v9.x level. Optionally, you could then export a new JT choosing JT version 9.1 or later.

By default, associative measurements have a different color scheme than non-associative measurements, with green edges and a white background fill, which makes them easier to identify in the 3D Viewing window. To display non-associative measurement labels in the same color scheme as standard measurement labels, on the **Display** page of the **3D measurement preferences**, clear the **Use fixed color scheme for associative measurements** check box.

Circle
Center: [6.55 -0.00 -21.93] mm
Diameter: 12.00 mm
Length: 37.70 mm

Associative measurement

Circle
Center: [6.55 -0.00 -21.93] mm
Diameter: 12.00 mm
Length: 37.70 mm

Non-associative measurement (default colors)

When you open a session file, if the **Restore during session load** measurement preference is active, saved associative measurements initially appear as *restored* measurements (the same as non-associative measurement). If the **Update during session load and snapshot apply** measurement preference is active, saved associative measurements update when you:

- Select **Measurement**.
- Apply a snapshot or 3d geometry asset with measurements.

- Open a PLM XML file that was saved with a view of associative measurements.

The table below summarizes the major differences between associative and non-associative measurements.

Category	Non-associative measurements	Associative measurements
History	Original measurement type	Introduced in Lifecycle Visualization 8.1
JT file support	No special JT data required	Require JT files with B-Rep Entity IDs If you have older JT files (JT v9.0 or earlier) with B-Rep Entity IDs, you may need to use Generate Selection Preview Data to update the part tessellation to a v9.x level. Optionally, you could then export a new JT choosing JT version 9.1 or later.
Appearance	Labels have cyan edge and gray fill color (by default).	Labels have green edge and white fill color
Updating	Cannot update	Update dynamically when their associated parts are transformed
Highlighting	Cannot highlight their associated entities after save and open	Can highlight their associated entities after save, open, and update
Clear (delete all)	Cannot be deleted with Clear after save and open	Can all be deleted with Clear after save, open, and update
Logging	Cannot be logged over time (since they never update)	Can be logged (double measurements only)
Label text	After save and open, always contain the label *** Restored Measurement *** (default value)	After save and open, contain the label *** Restored Measurement *** (default value) until updated Header text disappears after the update

Displaying information about active double measurement

Overview of displaying information about the active double measurement

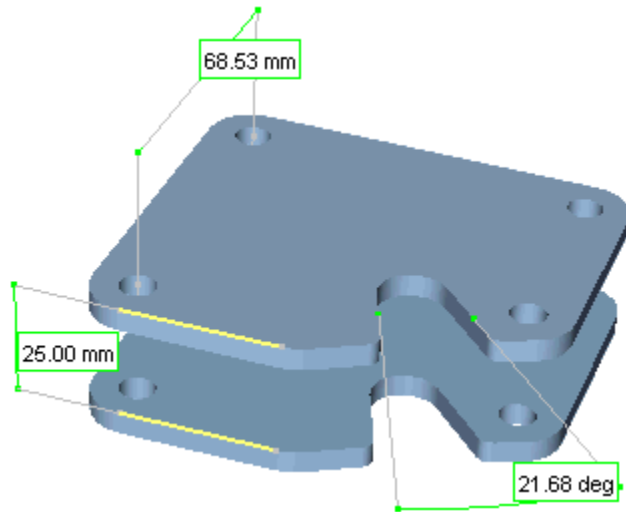
You can display information in the Viewing window about the active double measurement. For associative double measurements, the information updates dynamically if the parts are moved, rotated, or scaled.

ACTIVE MEASUREMENT DATA:

```

-----
ID : 25
Part1 : 1234_2
Part2 : 1234_1
Frame : 0
Distance : 25.00 mm
Pt1(x) : -37.653072
Pt1(y) : 46.011673
Pt1(z) : -5.000000
Pt2(x) : -37.653072
Pt2(y) : 46.011673
Pt2(z) : -30.000000
Dir(x) : 0.000000
Dir(y) : 0.000000
Dir(z) : -25.000000

```



The information includes:

- ID number of the measurement.
- Names of the two parts.
- Frame number of the motion file or transformation.
- Measured distance or angle.
- Coordinates of the two measurement points.
- XYZ delta between the two points.

Note:

- Only double measurements are displayed.
- If you delete the active double measurement, the data for the most recently created one is displayed (if one exists).

Display active double measurement information

Display information about the active double measurement in the upper left corner of the Viewing window.

1. Create at least one double measurement.
2. Choose **Measurement**→**Display Active Measurement**.

The Active Measurement Data for the most recently create double measurement appears in the upper left corner of the Viewing window.

3. (Optional) In the Viewing window, select a different measurement label to make it the active measurement.

The measurement data updates to reflect the measurement that you selected.

4. To turn off Active Measurement Data, choose **Measurement→Display Active Measurement** again.

Note:

- If the active measurement is associative, the information updates dynamically when either of the parts are moved, rotated, or scaled, either interactively or with a motion file.
- **Display Active Measurement** is disabled when there are no double measurements.
- If you select a measurement type other than double, no information is displayed.

Logging changes to measurements

Overview of logging changes to measurements

You can record initial information for all double measurements and log any subsequent changes to double **associative measurements** as you play a motion file or interactively move parts.

For example, you can measure the minimum distance between two parts, and play a motion file that moves and rotates the parts. Changes to the minimum distance are recorded as the parts move.

You can:

- Record information for all associative double measurements as they change over time.
- Start and stop logging without moving any parts to capture static information on all double measurements.
- Save the results to an HTML, CSV, TXT, or XML file.

Log measurement changes

Record the initial values for all double measurements and log any measurement changes as parts move.

Note:

Only **associative measurements** change as parts move.

1. Select **Measurement**→**Measurement Logging**→**Record**.
2. (Optional) Play a motion file or interactively move, rotate, or scale parts.
3. Select **Measurement**→**Measurement Logging**→**Stop**.

The measurement information appears in the **Report Definition Wizard** dialog box.

4. To save the report, click **Next**.
5. Specify a file format, path, and file name.

Tip:

To capture static information on all double measurements, start and stop **Measurement Logging** without moving any parts.

Working with measurement reports

Overview of working with measurement reports

Calculate measurements for multiple parts and surfaces, and generate reports that display results, optional properties, and attributes. You can edit your reports, change the density for your calculations, and then update your report. Export results as a text file, or save and retrieve your reports using session files.

Measurement reports have the following contents:

In this section	You can
Report List	View the titles of your measurement reports. Select a report, and then click Edit , Delete , Save , Export , or Recalculate . Click New to create a new report.
Description	Provide an optional description.
Single Results	View a table showing properties and attributes for each selected part. Each row represents a part. (A part name preceded by an asterisk (*) indicates some of the part's surfaces were not selected for the report.) The first column is Part Name . There is a column for each part property and attribute you specify.

In this section	You can	
	This type	Includes
	Calculated properties	The following for parts: • Area • BBox (bounding box) • Center of Mass • Mass • Volume Area for surfaces.
	Defined properties	• Density Source The value can be User Defined or Default. • Density The value is used for part mass calculations.
	Optional Attributes	The attributes for the part.
	Note: If you select one or more surfaces, Area is the only calculated property available.	
Density	Enter a new value for the density.	
Units	Specify mass and linear units.	
Summary	View the following: • Any error or warning messages about the calculations. • Totals of all calculated properties. • Moment of Inertia if included in the report. • Principle Axis of Inertia if included in the report.	

Generate measurement reports

1. On the **3D Measurement** toolbar, click **3D Measurement** .
2. (Optional) In the **Measurement Preferences** dialog box, specify the **density** preferences used for mass calculations.
3. On the **3D Measurement** toolbar, click **Reports** .

The **Create Report** dialog box appears, prompting you to select the parts or surfaces to include in the report.

4. Do the following:

To select	Do this
All visible parts	a. Hold Ctrl, and on the 3D Selection toolbar, click Part . b. In the Create Report dialog box, click Pick All Visible. Note: Wireframe parts are not selected when Pick All Visible is selected.
Individual parts	a. Hold Ctrl, and on the 3D Selection toolbar, click Part . b. In the Viewing window, select the parts. Note: Selected parts are highlighted. You can select a highlighted part again to deselect it. Wireframe parts cannot be selected to be part of the report.
All surfaces of all visible parts	a. Hold Ctrl, and on the 3D Selection toolbar, click Surface . b. In the Create Report dialog box, click Pick All Visible.
Individual part surfaces	a. Hold Ctrl, and on the 3D Selection toolbar, click Surface . b. In the Viewing window, select the part surfaces.

To select	Do this
	Note: Selected surfaced are highlighted. You can select a highlighted surface again to deselect it.

- Click **Next**.

The second **Create Report** dialog box appears.

- For **Report Name**, type a name for the report.
- From the **Available Properties** list, select the properties you want calculated, and then click **Add**.

The properties move to the **Visible Columns** list.

Note:

For parts, properties include **Area**, **BBox**, **Center of Mass**, **Density**, **Density Source**, **Mass**, and **Volume**. For surfaces, the only property is **Area**.

- From the **Available Attributes** list, select the attributes you want to be displayed in the report, and then click **Add**.

The attributes move to the **Visible Columns** list.

- (Optional) Select the **Principal Axis of Inertia** and/or **Moment of Inertia** check boxes.

Note:

The **Principal Axes of Inertia** and **Moment of Inertia** options are available only if all the parts in the model contain XT B-Rep data.

- (Optional) To modify the order in which the properties and attributes appear within the report, in the **Visible Columns** section, select a property or attribute, and then click **Up** or **Down**.
- Click **Finish**.

The measurement report is generated.

Note:

- If you want to change your parts and surfaces selections, click **Back** before generating the measurement report.
- The report will contain a warning message if the following types of problems occur during calculation:
 - Any selected part has a geometric problem.
 - Required properties are not appropriate for the selected part(s).
- Not all properties and attributes are available for all types of data. For information on including metadata in parts, consult your CAD and translator documentation.
- Moment of Inertia results are displayed in this order: x, y, z.
- Inertia results will be invalid if the selected parts have a mixture of solids and sheets. To get valid results, parts must be the same type.

Regenerate a measurement report after modifying individual part densities

For measurement reports that include mass calculations, you can change individual part density values and regenerate the reports.

1. Generate a measurement report.
2. On the **3D Measurement** toolbar, click **Reports** .
3. From the **Report List**, select the name of the report to modify.
4. In the **Single Results** section, select the parts for which you want to modify the density value.
5. For **Density**, type a new value.
6. (Optional) Select other parts and modify their density values.
7. Click **Recalculate**.

The new results appear in the **Single Results** section.

Save measurement reports to a session file

1. On the **3D Measurement** toolbar, click **Preferences** .

2. In the **Measurement Preferences** dialog box, select the **General** tab.
3. In the **Measurement Reports** section, select the **Save to session file** check box.
4. Click **OK**.
5. Generate a measurement report.
6. Choose **File**→**Save Session**.
7. In the **Save As** dialog box, enter a name for the session file.
8. Browse to the appropriate directory, and click **Save**.

A session file containing the measurement reports is saved. All calculated results associated with each report are saved in the file.

Load measurement reports saved to a session file

1. Open a session file that contains a measurement report.
2. On the **3D Measurement** toolbar, click **Preferences** .
3. On the **3D Measurement** toolbar, click **Reports** .
4. In **Report List**, select a measurement report.

The measurement report is loaded.

Export a measurement report as a text file

1. Generate a measurement report.

Note:

You cannot load reports that have been exported as a text file. To preserve reports for later viewing or editing, **save the reports to a session file**.

2. On the **3D Measurement** toolbar, click **Reports** .
3. On the Report page, click **Export**.
4. In the **Save As** dialog box, enter a name for the text file.

5. Browse to the appropriate directory, and click **Save**.

The measurement report is saved as a text file with comma separated values.

Edit measurement reports

1. Generate a measurement report.
2. On the **3D Measurement** toolbar, click **Reports** .
3. In **Report List**, select the name of the report you want to edit, and then click **Edit**.
4. In the **Edit Report** dialog box, do any of the following:

To	Do this
Change the report name	In Report Name , enter a new name for the report.
Add or remove properties	In Visible Properties , select a property, and click Add or Remove .
Add or remove attributes	In Visible Attributes , select an attribute, and click Add or Remove .
Reorder existing properties or attributes	In Visible Properties , select a Property or Attribute, and then click Up or Down .
Enable or disable Principal Axis of Inertia	Select or clear the Principal Axis of Inertia check box. Note: The Principal Axes of Inertia and Moment of Inertia options are available only if the model contains XT B-Rep data for all selected parts.
Enable or disable Moment of Inertia	Select or clear the Moment of Inertia check box. Note: The Principal Axes of Inertia and Moment of Inertia options are available only if the model contains XT B-Rep data for all selected parts.

5. Click **OK**.

The modified report appears.

Note:

Recalculation occurs only if a new property is added.

Delete measurement reports

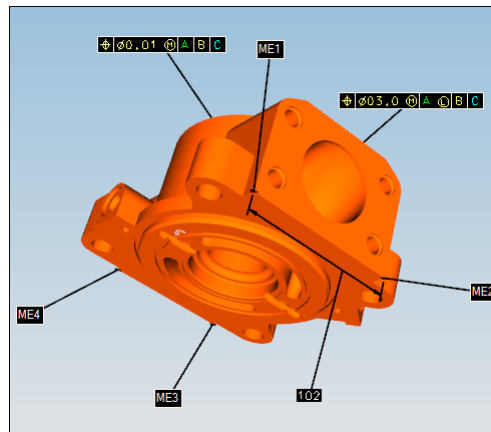
1. On the **3D Measurement** toolbar, click **Reports** .
2. In the **Report List** section of the **Reports** dialog box, select a report, and then click **Delete**.

The report is deleted.

Viewing Product and Manufacturing Information (PMI)

Introduction to viewing Product and Manufacturing Information (PMI)

You use the viewing application to display PMI when the PMI is generated by a CAD program and translated to a JT file. Your JT file may include several different types of PMI, for example, dimensions, notes, coordinating systems, and so forth. You can organize how PMI is displayed in several different ways. For example, you may need to only display measurement PMI, or just specific notes about the model. Or, you may want to examine specific sections of the assembly by using model views. These and several other methods are available when working with PMI.



PMI data is created in the CAD program and can be organized by PMI groups. The most common PMI groups include annotations, user attributes, design groups, and model views.

Note:

Beginning with version 10 of the JT format, Teamcenter Visualization supports generic PMI data only. Legacy PMI data (non-generic style) is not displayed. If you have PMI data in a non-generic style, convert it to generic data in your CAD application before translating your model to JT.

The following information briefly describes how PMI is organized, how it is represented graphically, and how you can interact with other software features when displaying PMI.

PMI groups

When PMI data is created in the CAD program it can be organized by PMI groups.

Symbol information

The display of different PMI data types is distinguished by graphic representation or symbols.

Interaction with other features

You can display individual PMI entities by using the **3D Selection** toolbar options. You can choose the entity (point, vertex, edge, surface, or part) before you display the PMI.

Note:

- In the **PMI** view, a check mark is placed in the **Has PMI** column when the assembly or part contains PMI.
- For Teamcenter, viewing PMI is supported by several views, for example, Lifecycle Viewer, Manufacturing Process Planner, and Structure Manager.

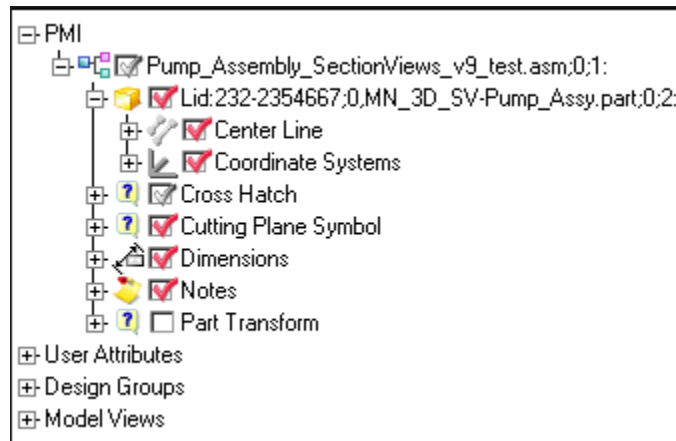
Overview of the PMI view

In the **PMI** view, the PMI for the selected part or assembly is displayed in a hierarchical tree. PMI is organized by type or section and each translated model contains up to four different PMI sections. Remember that the viewer simply displays PMI that has been translated from the native CAD data.

The four PMI sections are:

- *PMI*. This section contains individual PMI annotations, grouped by PMI type. You can display or hide individual annotations, all annotations of each type, or all the PMI annotations at once.
- *User Attributes*. This section contains metadata such as the Bill of Materials, Change History, Description, and Weld Attributes PMI data, if they are available for the selected part or assembly. Metadata is displayed as text.
- *Design Groups*. This section contains the design groups of part geometry and PMI as they were created in the CAD program. You can display only one design group at a time. For instance, when you select the status indicator to the left of the design group name, that design group appears, but all other PMI data and design groups are not displayed in the Viewing window.
- *Model Views*. This section contains the model views of surfaces, PMI, and the model view created in the CAD program. If the model view includes individual entities (surfaces, edges, vertices, or points)

instead of the entire part, only those entities are displayed. You can display only one model view at a time.



You can display or hide individual PMI or sets of PMI as they are listed in the view. Displaying some groups of PMI hides the display of other PMI. The following table explains how PMI is displayed or hidden.

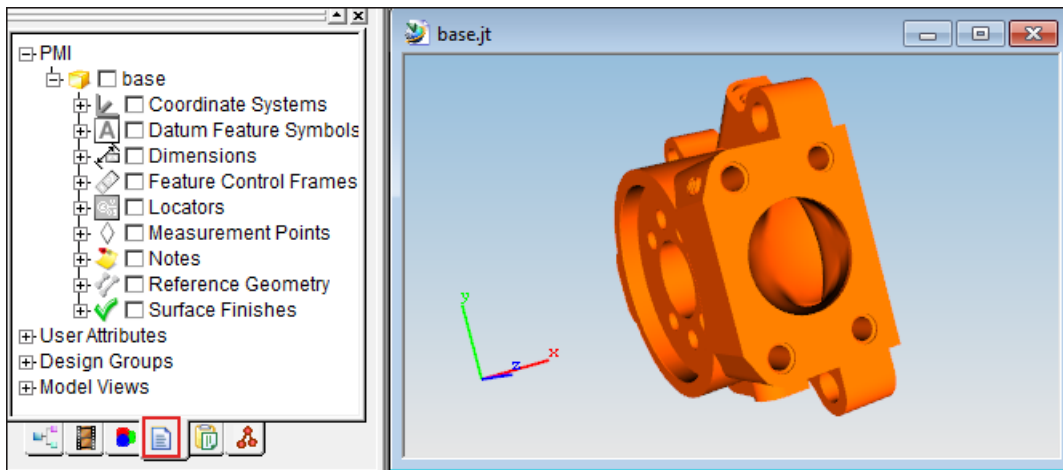
Note:

- The state indicators are gray for sections that include multiple **Set Geometry Visibility** PMI options. This happens because **Set Geometry Visibility** displays the entities one group at a time.
- You can associate several GD&T entities so that when you make one of them visible in the **PMI** view, all the associated entities become visible. These linked entities have separate entries in the **PMI** view, but they appear as a group in the Viewing window.
- When you select an icon in the **PMI** view, you select the associated PMI label and surfaces in the Viewing window. When you select a PMI label in the Viewing window, you select the associated surfaces and the icon in the **PMI** view.

PMI Annotations

You can display all or just selected PMI contained in the translated model.

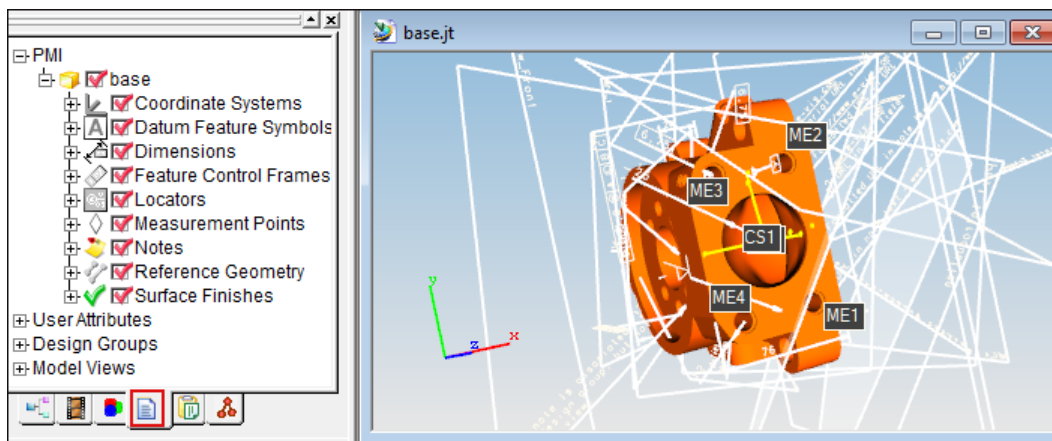
When you first turn on PMI, the list of PMI is displayed in the PMI view, but PMI is not yet visible in the Viewing window.



Use the PMI view to turn on different types of PMI. In the example below, **Dimensions** and **Measurement Points** have been turned on.

Note:

When you display all PMI associated with a model, the Viewing window might display hundreds of PMI. The result may not be useful.



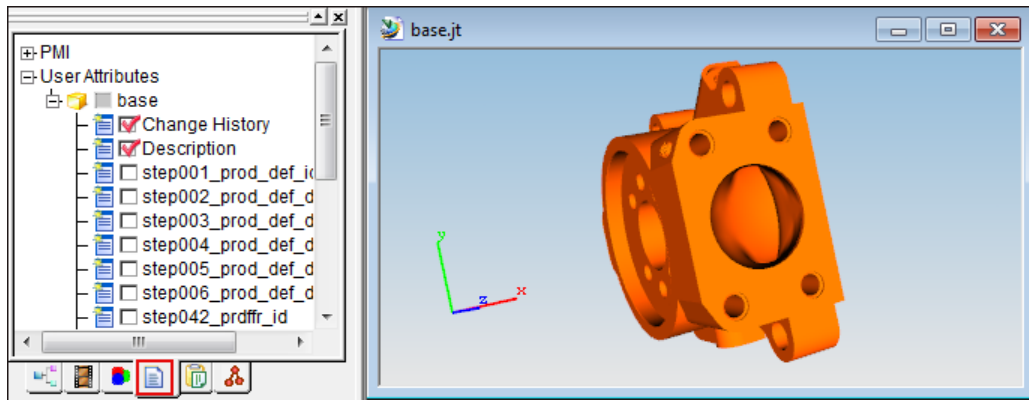
User Attributes

In the **PMI** view, the PMI is arranged into groups that were defined in the CAD program. User attributes, such as **Bill of Materials**, **Change History**, **Description**, or **Weld Attributes** are listed under the **User Attributes** group. You can display user attributes by expanding the **User Attributes** group and selecting an attribute name.

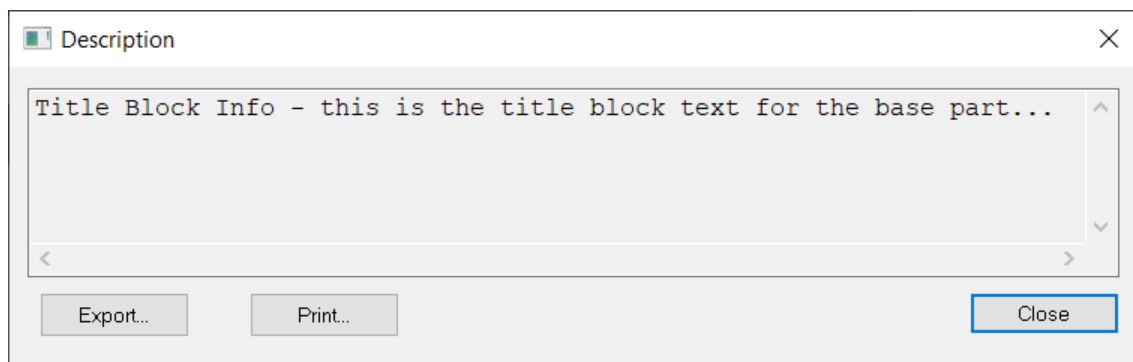
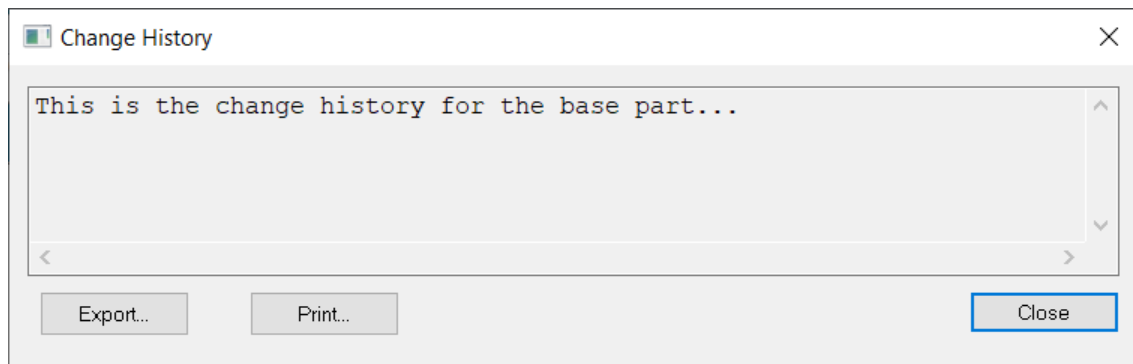
User attributes contains metadata, if available, that was created in the CAD program.

User attributes are displayed in text boxes. You can export and print the text, but you cannot edit or delete the text.

You can select all user attribute PMI by selecting each user attribute. You can also select one or more user attributes. For example, you can select **Change History** and **Description**.

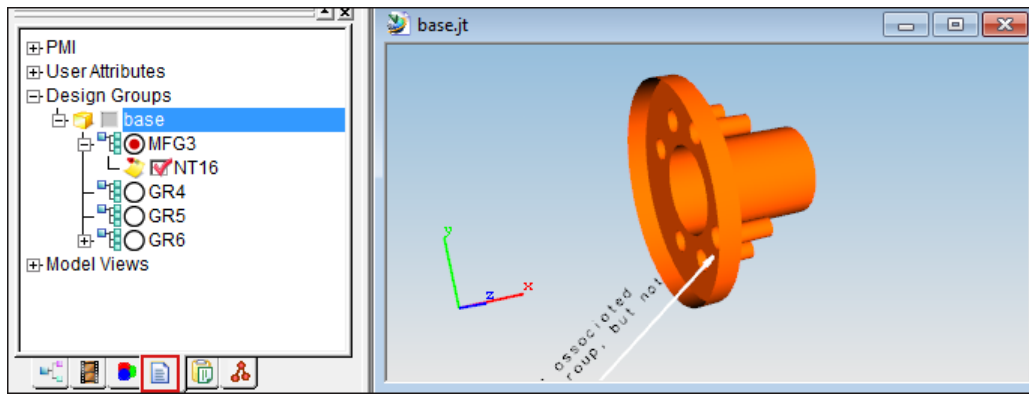


The PMI is displayed in individual text boxes.



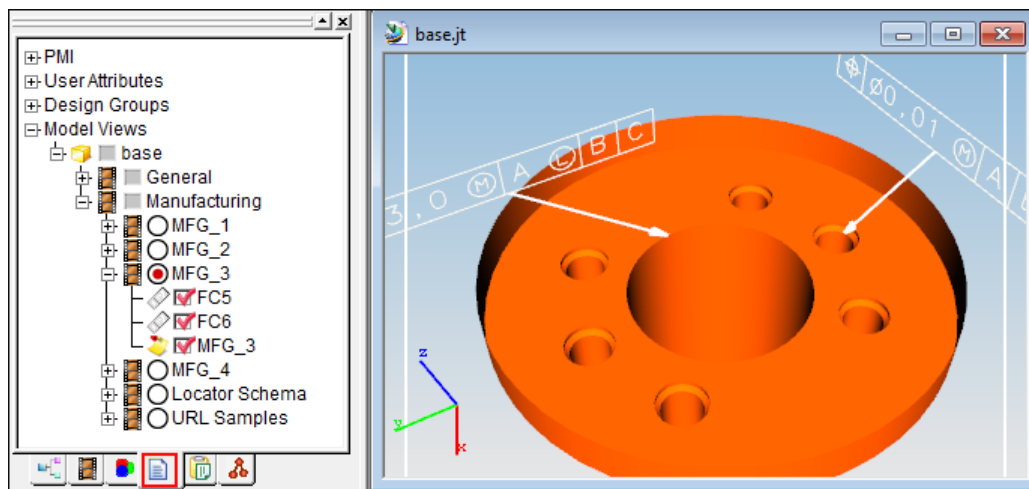
Design Groups

Design groups contain part geometry and PMI as they were created in your CAD program. As noted before, you can display only one design group at a time.



Model Views

Model views contain model surfaces as they were created in your CAD program. If the model view includes individual entities (surfaces, edges, vertices, or points) instead of the entire part, only those entities are displayed. As noted before, you can display only one model view at a time.



PMI types

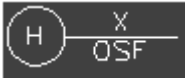
Overview of common PMI types

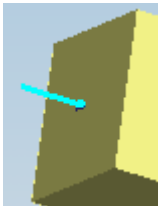
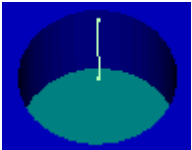
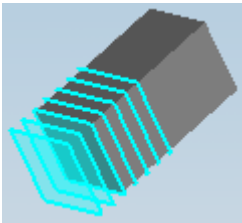
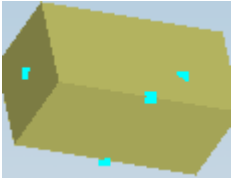
PMI types include the following:

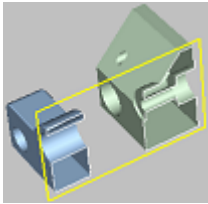
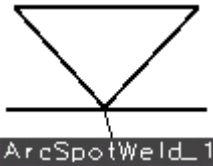
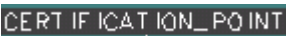
- *Notes* are text annotations that are attached to a part or entity and can contain any information.
- *Measurement Points* are geometric or theoretical points that can be measured on manufactured parts to verify the accuracy of the manufacturing process.
- *Surface Finish Symbols* identify the surface finish of a part or surface, if the finish quality affects the part function.

PMI symbols may differ slightly from those shown here. This following information lists the most common PMI types.

Common PMI types

Symbol Type	Symbol	PMI displayed
Coordinate Systems		User-defined coordinate systems in a particular part model.
Datum Feature Symbols		Label for a part feature, which is referenced by a feature control frame.
Datum Locator Symbols		Generic datum location area and control direction. This area is later refined for individual parts and subassemblies and becomes the specific datum points. May include parameters for vector direction, body section, and connected part information.
Datum Target Symbols		Specific point, line, or area on a part that defines a datum.
Datum Points		Type and application of points on a part that are used to consistently and accurately locate components with respect to each other and the manufacturing tooling. The datum point parameters include location, name, vector direction, and other parameters specific to their application.
Dimensions		Distance between geometric items.
Feature Control Frames (GD&T)		Symbols, including Datum Feature and Datum Target, that communicate the design intent for the part.
HTML Notes (Only for models created in I-DEAS)		Text notes that are hyperlinks. Follow the hyperlink in one of these ways: - Double-click the underlined URL in the HTML note. - Right-click the HTML note, and then select Display Web Page.
Measurement Locator Symbols		Generic measurement location area and control direction. This area is later refined for individual

Symbol Type	Symbol	PMI displayed
		parts and subassemblies and becomes the specific measurement points. May include parameters for vector direction, body section, and connected part information.
Measurement Points		Type of measurement point and predefined locations that may be either geometric entities or theoretical (but measurable) points. These points are measured on manufactured parts to verify the accuracy of the manufacturing process. The measurement point parameters include location, name, vector direction, and other parameters specific to their usage.
Mechanical Clinch Symbols		Mechanical clinch connector that is selected. This includes parameters for location, quantity, tolerance, and class.
Notes		Text that is attached to part entities (points, vertices, edges, etc.).
Reference Axis		The axis defined as reference geometry in the CAD package.
Reference Geometry		Points, lines, and planes defined as reference geometry in the CAD package.
Reference Plane		The plane defined as reference geometry in the CAD package.
Reference Point		The points defined as reference geometry in the CAD package.

Symbol Type	Symbol	PMI displayed
Section		Cross sections defined in the CAD package, including those authored as lightweight sections in NX 7.5 or later.
Structural Adhesive Symbols		Type and shape of structural adhesive being identified. The shapes may be bead, dollop, or tape, and the adhesive types sealer, mastic, or adhesive. Included in this symbol are parameters defining size, shape, and quantity.
Surface Finish Symbols		Surface finish, only if finish quality affects part function.
Welding Symbols		Type of weld and various other elements used to convey specific welding information. This includes contour, backing, all around, field weld, size parameters, and weld placement.
Certification Point Symbols		Certification points used to validate the machined surfaces in tooling fixtures. They help ensure proper contour and placement of NC locator blocks. The certification point-symbol identifies location, name, and vector direction. The points are generally grouped and named by the area they inspect.

Common PMI measurement types

The following is a list of common PMI measurement types:

- Surface Vectors
- Hole/Slot/Stud Vectors
- Hem Vectors
- Trim Vectors

Common PMI weld types

The following is a list of common PMI weld types:

- Groove Weld
- Fillet Weld
- Plug/Slot Weld
- Edge Weld
- Arc Spot Weld
- Resistance Spot Weld
- Resistance Seam Weld

Display PMI in the viewer

Two shortcut menu options control how PMI displays in the Viewer. Both options are only supported when:

- PMI is activated.
- You right click a part in the Assembly page or in the Viewing window.

Note:

PMI may not appear if the model was saved in a Teamcenter product view and then the dimension of the object was changed in a CAD program in a way that changed the PMI. When this occurs, the caption **PMI Missing** appears for any PMI that cannot be restored. (The text of this caption may differ depending on how the caption is defined on the **Snapshots** tab in the **PMI Preferences** dialog box.) Also, you may need to zoom out to see the caption.

1. Open an assembly containing PMI.
2. From **View→Toolbars**, choose **3D PMI**.
3. On the **3D PMI** toolbar, select **PMI**.
4. Right-click a part and then select one of the following options:
 - Right-click a part and select **Show PMI**.

This option displays all PMI on a part or subassembly, including their children.

- Right-click a part and select **Toggle in PMI Tree**.

This option adds all configured PMI from a part or subassembly into the PMI page.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page when either **Show PMI** or **Toggle in PMI Tree** is selected.

Determining how you want PMI displayed

Selecting options for how you want to display PMI

You can choose how you want PMI displayed in the Viewing window. For example, in some cases you need to view all PMI associated with the model. At other times you need to review and display specific types of PMI, or PMI associated with the design group of the model view that was created in the CAD program.

Optionally, to optimize how PMI is displayed, you may want to change the font style and size, the color of dimension geometry, or change how labels appear in the Viewing window.

Useful display options include:

- Turning on and displaying PMI.
- Sorting PMI.
- Displaying PMI one entity at a time.
- Displaying one PMI entity or all associated PMI entities of the model.
- Displaying PMI associated with **User Attributes**.
- Displaying PMI associated with **Design Groups**.
- Displaying PMI associated with **Model Views**.
- Displaying specific PMI types by using filters.
- Displaying PMI effectivity controlled views.

Note:

If you select a part in the assembly and then turn on PMI, only the PMI associated with that part is displayed.

Activate PMI features

If the model is new or you are unsure the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

1. On the **3D PMI** toolbar, select **PMI**.
2. Right-click the model in the Viewing window and choose **Toggle in PMI Tree**.

When you select **Toggle in PMI Tree**, you control how PMI is displayed in the Viewing window.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

If you first select a part and then turn on PMI, you only see the PMI of the selected part. To see all PMI of the model, do not select a part.

Sort PMI

You can organize the PMI displayed in the **PMI** view.

1. On the **3D PMI** toolbar, select **PMI**.
2. Right-click any PMI entity and choose **Sort**.

The **Sort Tree** dialog box appears.

3. Perform one of the following:
 - Select **Override PMI Sort Order in the File** and specify the sorting preferences in the dialog box.
 - Clear the selection of **Override PMI Sort Order in the File** to use the sorting preferences from NX.

Note:

PMI data must come from NX 9.0 or later.

4. Click **OK** to close the dialog box.

Display PMI one entity at a time

1. On the **3D PMI** toolbar, select **PMI**.

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

3. On the **PMI** toolbar, click **Single Entity Mode** .
4. To display the next PMI for the object, from the **PMI** toolbar, click .
5. To move backward and review previous PMI, click **Previous Entity** .

Display one or more PMI entities

1. On the **3D PMI** toolbar, select **PMI**.

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

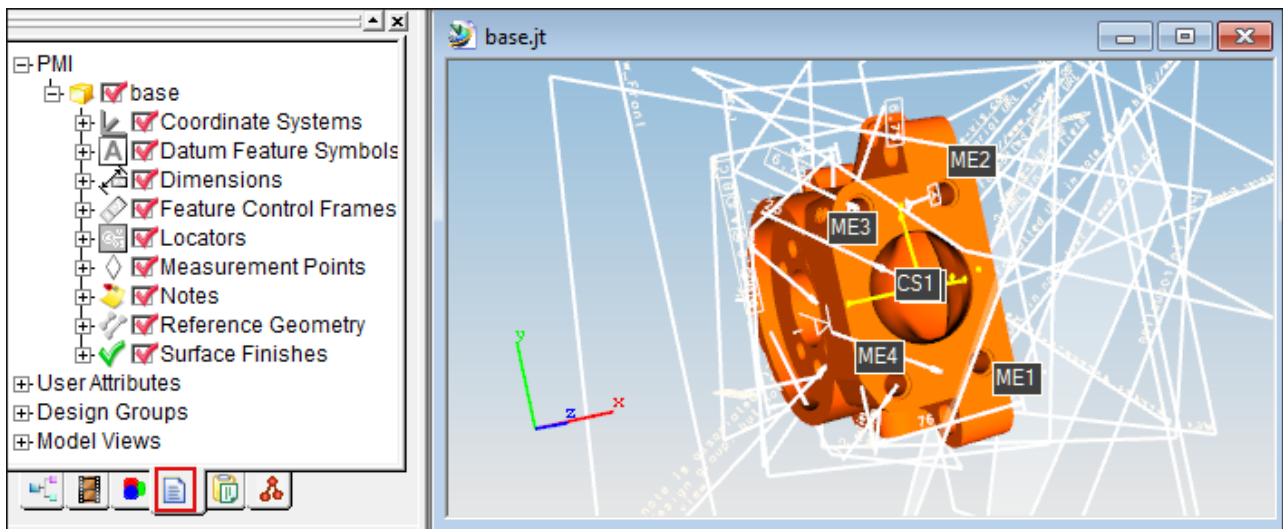
2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

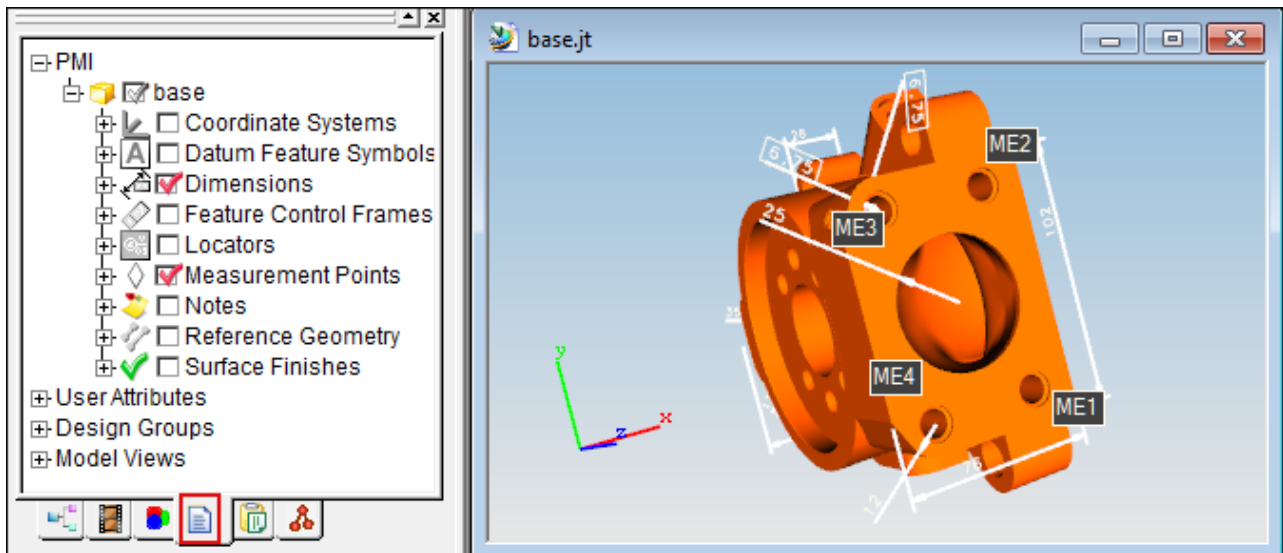
Note:

- If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog box, the view is automatically switched to the PMI page.
- If **Automatically populate PMI tree** is selected on the **PMI Preferences** dialog box, you do not need to complete this step.

3. In the **PMI** view, expand **PMI**.
4. Select the top node or base of the model and all PMI associated with the model is displayed in the Viewing window.



5. Clear the top node, in this case **base**, and then select one or more PMI entities to display in the Viewing window.



- To view the properties of each PMI entity, in the **PMI** view, right-click the PMI and choose **Properties**.

The **PMI Properties** dialog box appears, displaying data about the PMI entity.

Display PMI with user attributes

- On the **3D PMI** toolbar, select **PMI**.

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
base	✓

- Right-click the model and choose **Toggle in PMI Tree**.

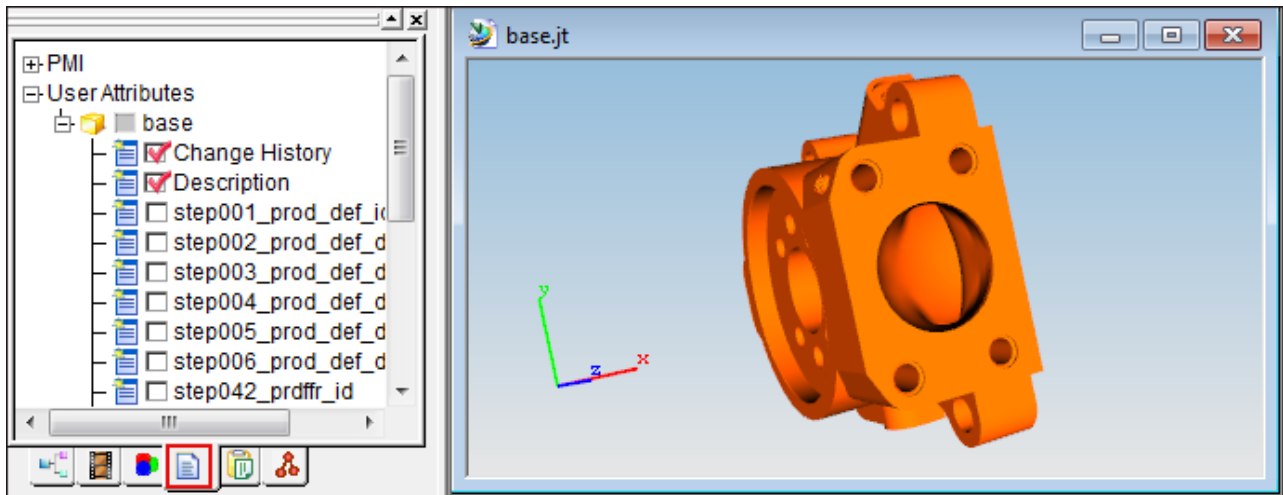
If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

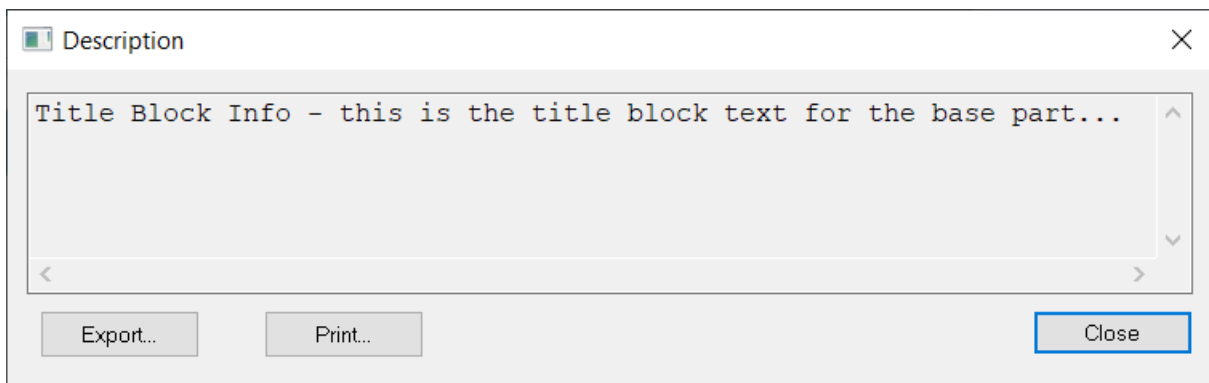
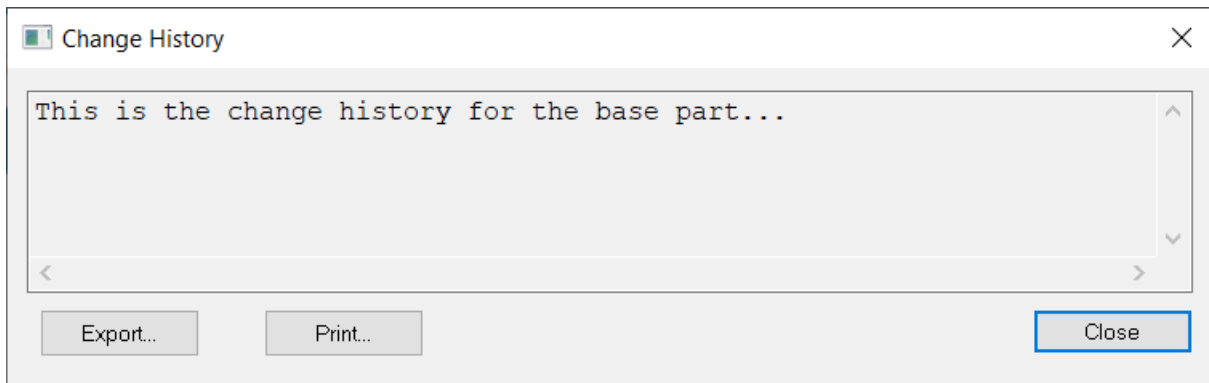
If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

- In the **PMI** view, expand **User Attributes**.

4. Select types of user attributes, such as **Change History** and **Description**, then review the text information about them.



User Attributes PMI is displayed in individual text boxes.



Display PMI using design groups

1. On the **3D PMI** toolbar, select **PMI**.

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

- Right-click the model and choose **Toggle in PMI Tree**.

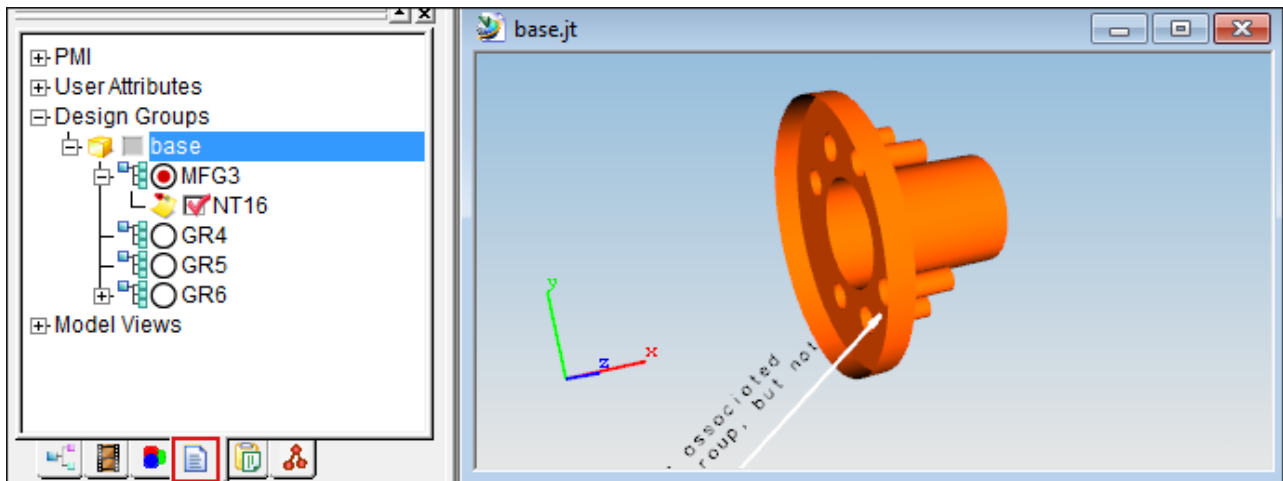
If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

- In the **PMI** view, expand **Design Groups**.
- Select any one design group to view its structure in the Viewing window. If the design group contains PMI that is also displayed.

You can only display one design group at a time.



- To organize how design groups are displayed, right-click anywhere in **Design Groups** and choose from the following:

Use this option	To do this
Set Geometry Visibility	Display the entities (parts, surfaces, edges, vertices, and points) for this design group only. All other part geometry is hidden.
Set PMI Visibility	Display the PMI for this design group. The display of other PMI is not affected.
Sort Alphabetically	Sort design groups alphabetically within its group. The PMI tree structure for other entities is not affected.
Properties	Display design group properties.

Display PMI using model views

1. On the **3D PMI** toolbar, select **PMI**.

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

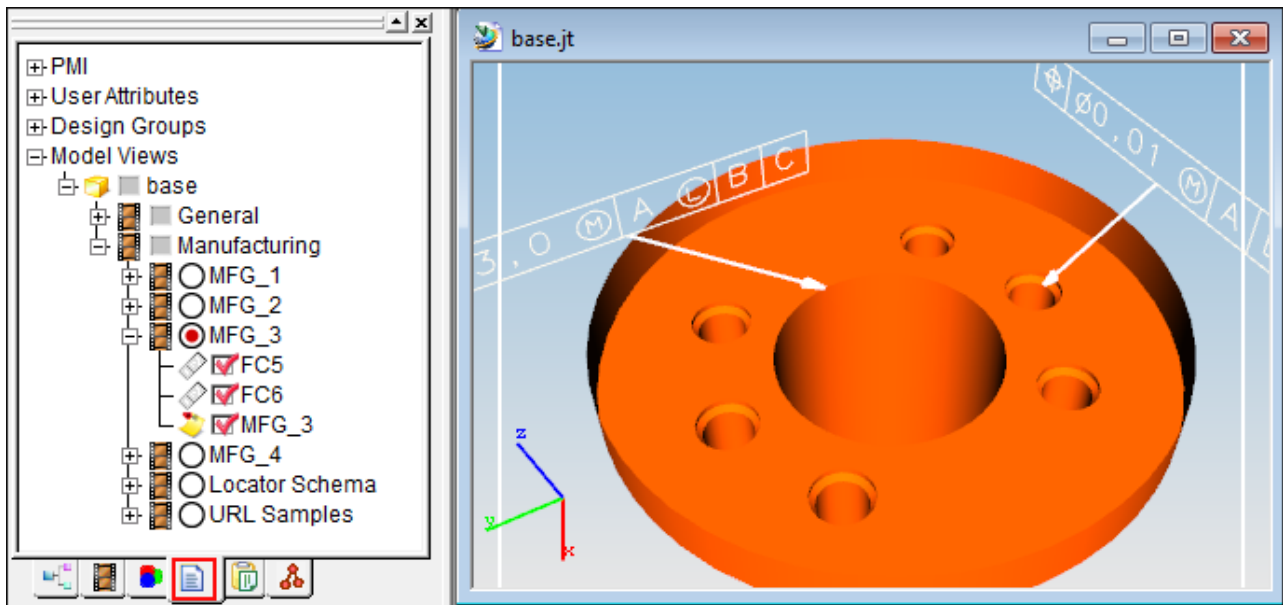
Item Name	Has PMI
Models	
└─ base	✓

2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

3. In the **PMI** view, expand **Model Views**.
4. Select a model view and display it in the Viewing window.

You can only display one model view at a time.



5. To organize how design groups are displayed, right-click anywhere in **Model Views** and choose from the following:

Use this option	To do this
Merge Model View	Merge PMI from another model view into the active model view. - Select a Model View. - Right-click another Model View and choose Merge Model View.
Set View Navigation	Change the view of the model and PMI in the Viewing window to the view specified by another model view. The display of model parts and PMI is not affected. - Select a Model View. - Right-click another Model View and choose Set View Navigation.
Set Geometry Visibility	Display the entities (parts, surfaces, edges, vertices, and points) for this model view only. All other part geometry is hidden.
Set PMI Visibility	Display the PMI for this model view. The display of other PMI is not affected.

Use this option	To do this
Sort Alphabetically	Sort model views alphabetically within its group. The PMI tree structure for other entities is not affected.
Properties	Display model view properties.

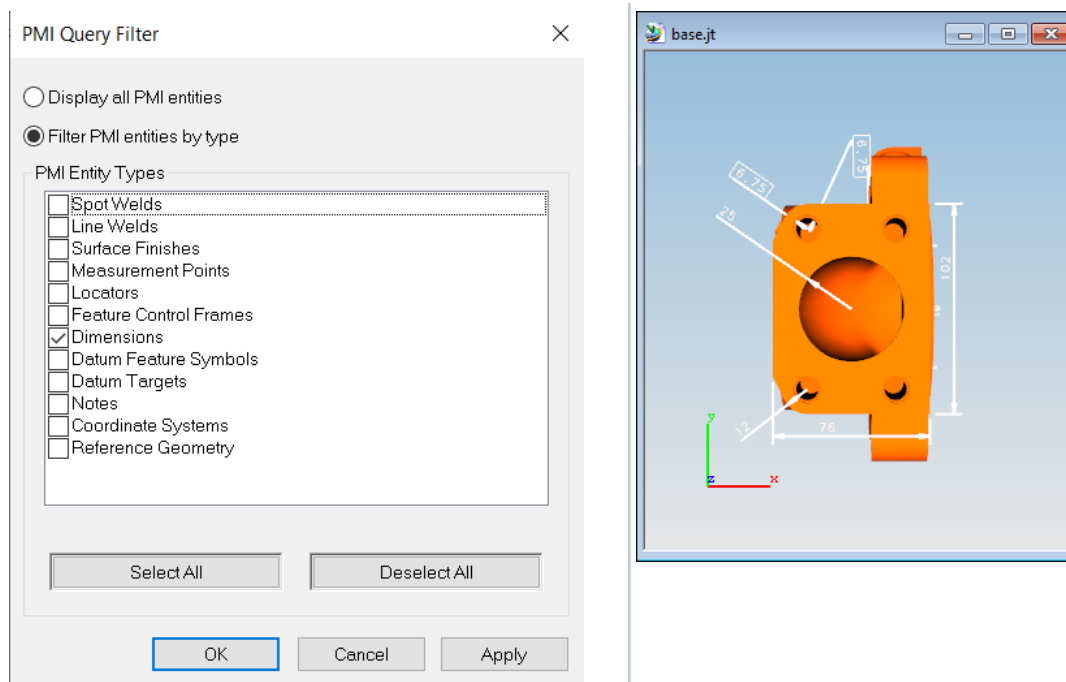
Display PMI types using filters

Use PMI filters to specify the PMI entities you want to display in the Viewing window.

1. On the **PMI** toolbar, click **Filtering** .
2. Choose **Filter PMI Entities by Type** from the **PMI Query Filter** dialog box.
3. To display or hide each PMI type, select or clear **PMI Entity Types**.
4. Click **OK**.

Example:

In this example, you only want to display **Dimensions** PMI.



Display PMI using specific 3D selections

1. On the **PMI** toolbar, click **Filter Configuration** .

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

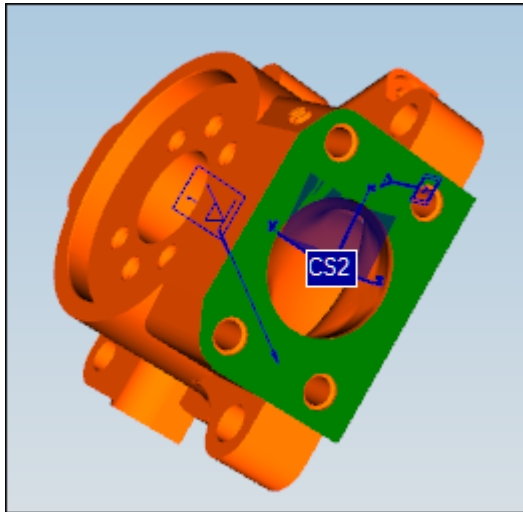
3. On the **PMI** toolbar, click **Multiple Entity Mode** .
4. On the **3D Selection** toolbar, click one of the following:
 - **Edge** 
 - **Surface** 
 - **Part** 

5. In the Viewing window, select the corresponding entity, for example a surface.

All associated PMI and B-Rep is displayed.

Example:

If you clicked **Surface**  in step 4 above and then selected a surface in the Viewing window, the PMI and B-Rep associated with the surface is displayed.



Display PMI using feature level designs

1. On the **PMI** toolbar, click **Filter Configuration** .

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

3. On the **PMI** toolbar, click **Feature-based Multiple Entity Mode** .
4. If the model contains feature level PMI, select any element of the feature level.

PMI for this group is displayed.

Example:

A model contains a feature level of a cylinder and its endcaps. When you select either the cylinder or an endcap, you display the same PMI. If the model does not contain feature levels, PMI for the cylinder is displayed and PMI for the endcap is displayed.

If the selected part or section does not contain PMI or if you have already activated PMI in the area, a message appears that new PMI is unavailable.

Display PMI using multiple entities

1. On the **PMI** toolbar, click **PMI** .

Note:

If you are unsure that the model includes PMI, look for a green check mark under the **Has PMI** column in the **Assemble** view.

Item Name	Has PMI
Models	
└─ base	✓

2. Right-click the model and choose **Toggle in PMI Tree**.

If you have a part selected during this step, you will only see the PMI of the selected part.

Note:

- If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog box, the view is automatically switched to the PMI page.
- If **Automatically populate PMI tree** is selected on the **PMI Preferences** dialog box, you do not need to complete this step.

3. On the **PMI** toolbar, click **Multiple Entity Mode** .
4. On the **3D Selection** toolbar, click one of the following:
 - **Edge** 
 - **Surface** 

- Part

5. In the Viewing window, select the corresponding entity, for example a surface.

All PMI associated with the selected entity (edge, surface, or part) is displayed. Continue selecting entities, for example, surfaces. If PMI exists, it appears in the Viewing window.

If the selected entity does not contain PMI or if you have already activated PMI in the area, a message appears that new PMI is unavailable.

Working with model views

Overview of model views

Model views are 3D representations of product components created in NX. Model views published by NX to Teamcenter and to JT files are attached to a disclosure object in Teamcenter. This object is referred to as a *3D design disclosure* and serves as the 3D model based definition for a design.

This feature is supported with NX 11.0.1 and later.

Setting up model views in NX and Teamcenter

Required setup for model views

For model views to be available in Teamcenter and Teamcenter lifecycle visualization, you must set up the following:

- **Publish model views** to Teamcenter from NX.
- **Configure Teamcenter** to display **model view objects**.
- In either the **rich client** or **Active Workspace**, add the **Disclosure** tab.

Publish model views for disclosure in NX

After creating and arranging PMI in model views in your assembly or model you may want to limit which views can be used during design reviews, planning processes, or other official types of collaboration with customers or suppliers. In other words, you may want to limit which model views are published for use by others.

In NX, two model view property attributes let you indicate which model views may be used for collaboration, and for what purpose.

- The **Publish for Disclosure** model view property attribute lets you designate which model view in your part or assembly can be published to Teamcenter for collaboration.

- The **Disclosure Purpose** model view property attribute lets you identify the purpose for disclosing the model view. For example, the purpose can be to disclose the detailed design definition for a given part, assembly, or installation as a collection of model views attached to a disclosing object.

You can set these attributes for each model view individually, or you can select one or more model views and collectively set their attributes using a bulk edit method.

Once these attributes are set, they can be referenced by Teamcenter lifecycle visualization to manage which views are presented as a 3D design definition.

Refer to the NX help for information about publishing model views for disclosure.

Enable JT dataset creation from NX

To facilitate JT datasets at all levels of the structures and hierarchies to be disclosed, you must enable multi-CAD to see assembly level PMI in Teamcenter lifecycle visualization.

On the NX client, set this environment variable:

```
UGII_GENERATE_MULTI_CAD_JT=1
```

It is recommended that this environment variable be added to the local application startup scripts for NX. For example, within the UGII_ENV.DAT file.

Refer to the NX help for more information about multi-CAD and about publishing model views for disclosure.

Overview of model view objects in Teamcenter

The model-view-related objects in Teamcenter are:

- **ModelViewProxy**

An object representing a proxy to the model-view-related object defined in the CAD model. Model views published from NX to Teamcenter are stored as ModelViewProxy objects in Teamcenter. ModelViewProxy objects are owned by the ItemRevision that owns the CAD model.

- **Disclosing Object**

An object that collects all geometry necessary to create all the model views required to define a design disclosure (for example, the detailed design definition) for a part, assembly, or installation. This object is sometimes referred to as a background collector, because it gathers all foreground and background geometry necessary to create the model views fully describing the design. Example object types are regular BVR assemblies, or Worksets. This object owns a ModelViewPalette.

- **ModelViewPalette**

An object that collects a list of ModelViewProxy objects for a given Disclosing Object. This object is related to a Disclosing Object by a Disclosed View List relationship.

Display of model view related objects in Teamcenter is controlled by **configuration**.

Display model view objects in Teamcenter

Display of **model view related objects** in Teamcenter is controlled by configuration. Follow these steps to display model view objects in Teamcenter clients.

1. Choose **Edit→Options**.
2. Expand the **Options** tree and expand the **General** folder.
3. Select the **Item Revision** node.
4. On the **General** options tab, select each of the following from the **Available Relations** list, and click **+** to move them to the **Shown Relations** list.
 - **Owned Model Views**
 - **Disclosed View List**
 - **Disclosing Object**
5. Click **Apply**.
6. Group the owned model views in a pseudofolder.
 - a. On the **Related Object** tab, select **Owned Model Views** from the **Available Relations** list, and click **+** to move it to the **Shown Relations** list.
 - b. Click **Apply**.
7. Repeat the above steps for all appropriate owning object and disclosing object types.
8. Click **OK**.

Add the Disclosure tab to the rich client Summary view

You can modify the **Summary** view in Teamcenter to add a **Disclosure** tab to display information about model views. The **Disclosure** tab can also include a button to open a model view in Teamcenter lifecycle visualization. You do this using XML rendering templates (XRTs).

1. Log on to Teamcenter as an administrator.

2. In Teamcenter preferences, search for the preference *business_object_name.SUMMARYRENDERING*.
3. Note the **Value** of the preference. This is the dataset name. For example: **Scp0ItemRevisionSummary**.
4. In **Advanced Search**, perform a **Dataset** search (system-defined search) using the dataset name.
5. Open the XML stylesheet attached to the dataset, and edit the stylesheet by adding this section in the appropriate location based on where you want the tab to appear in the view:

```

<page title = "Disclosure" titleKey = "">
= "" title = "Disclosed Model Views">
"wnd0DisclosedModelViewData.Fnd0DisclosedModelViewData"
sortdirection = "ascending" sortby = "wnd0DisclosedModelView"
defaultdisplay = "tableDisplay">
<tableDisplay>
"wnd0ModelViewPalette"/>
"wnd0ModelViewGroup"/>
"wnd0DisclosedModelView"/>
= "wnd0OwningModel"/>
= "wnd0OwningUser"/>
"wnd0OwningGroup"/>
<treeDisplay>
"wnd0ModelViewPalette"/>
"wnd0ModelViewGroup"/>
"wnd0DisclosedModelView"/>
= "wnd0OwningModel"/>
"wnd0OwningUser"/>
"wnd0OwningGroup"/>
<thumbnailDisplay/>
<command actionKey = "openInVisAction"
commandId = "com.teamcenter.rac.openInVMU" renderingHint =
"commandbutton"/>
</page>

<section titleKey =
<objectSet source =
"wnd0DisclosedModelViewData"
sortdirection = "ascending" sortby = "wnd0DisclosedModelView"
defaultdisplay = "tableDisplay">
<tableDisplay>
<property name =
<property name =
<property name =
<property name =
<property name =
<property name =
</tableDisplay>
<property name =
<property name =
<property name =
<property name =
<property name =
</treeDisplay>
<listDisplay/>
<command actionKey = "openInVisAction"
commandId = "com.teamcenter.rac.openInVMU" renderingHint =
"commandbutton"/>
</objectSet>
</section>
</

```

6. Repeat the above steps for all object types for which you want to add the **Disclosure** tab.
7. Add the button to open a model view in Teamcenter lifecycle visualization from the Disclosure tab:
 - a. Choose **Edit→Options**.
 - b. Expand the **Options** tree and expand the **Visualization** folder.
 - c. Select the **Lifecycle Visualization** node.

- d. Select **Yes** for **Show "Open" in Lifecycle Visualization command**.

Add the Disclosure tab to Active Workspace

You can modify the view in Active Workspace to add a **Disclosure** tab to display information about model views. You do this using XML rendering templates (XRTs).

Following are basic instructions for adding the tab. Refer to the Active Workspace help for more information on:

- Considerations for using XRTs in Active Workspace
 - Active Workspace-specific style sheets
 - Active Content configuration tasks
1. Log on to Teamcenter as an administrator.
 2. In Teamcenter preferences, search for the preference **AWC_business_object_name.showObjectLocation.SUMMARYRENDERING**.
 3. Note the **Value** of the preference. This is the dataset name. For example: **Awb0ItemRevSummaryForShowObjectLocation**.
 4. In **Advanced Search**, perform a **Dataset** search (system-defined search) using the dataset name.
 5. Open the XML stylesheet attached to the dataset, and edit the stylesheet by adding this section in the appropriate location depending on where you want the tab to appear in the view:

```
<page title = "Disclosure" titleKey = ""> <section titleKey
= "" title = "Disclosed Model Views"> <objectSet source =
"fnd0DisclosedModelViewData.Fnd0DisclosedModelViewData"
defaultdisplay = "tableDisplay" sortBy =
"fnd0DisclosedModelView" sortdirection =
"ascending"> <tableDisplay> <property name
= "fnd0ModelViewPalette"/> <property name
= "fnd0ModelViewGroup"/> <property name
= "fnd0DisclosedModelView"/> <property name
= "fnd0OwningModel"/> <property name =
"fnd0OwningUser"/> <property name = "fnd0OwningGroup"/>
</tableDisplay> <thumbnailDisplay/> <listDisplay/> </
objectSet> </section></page>

</page>
```

6. Repeat the above steps for all object types for which you want to add the **Disclosure** tab.

Customize metadata stamping on model views

If PLM metadata was created on a disclosed model view in NX, it is printed with the model view to allow control of printed design content.

You can customize the print stamping through the **GetMetaDataStamp** user exit ITK extension.

For more information about user exits, refer to Teamcenter Server Customization in the Teamcenter help.

Update a model view's user state

Use this process to change the user state on a model view. To clear a user state, select **None**.

1. In the Model Views view, right-click a thumbnail image or a part in the model tree and select **Update User State**.

You can select multiple images by holding down the Ctrl key as you select images.

2. On the **Update User State** dialog box, do the following.
 - a. Click the **User Review State** drop down list and select the user state to be applied to the listed model views.
 - b. Click **OK**.

The user review state value changes in the **User State** column on the tree and in the **Properties** dialog box.

Create a live 3D section from a PMI section

Use the context menu in either the Model Views or PMI view to create a live 3D section from a PMI section. Unlike PMI 3D sections, live 3D sections can be deleted, repositioned, realigned, and have parts added or removed.

- In the Model Views or PMI view, from the list, right-click a PMI 3D section and select **Create Section**.

The view is automatically switched to the **Cross Sections** page.

Highlight model views for selected PMI

When you are viewing a 3D model with PMI, you can find and display the model views that contain selected PMI.

1. When you are viewing an assembly containing PMI, right-click a part and select **Show PMI**.

Note:

If **Automatically switch to PMI tab** is selected on the **PMI Preferences** dialog, the view is automatically switched to the PMI page.

2. Select the PMI you want to find the model views for.
3. Right-click the PMI, choose **Related Model Views**, and then choose one of the following:

- **Highlight in Tree**

The model views containing the selected PMI are highlighted in the PMI view.

- **Show View**

The model view containing the selected PMI is displayed in the Viewing window. If the PMI is contained in more than one model view, choose **Show View** again to cycle through the model views.

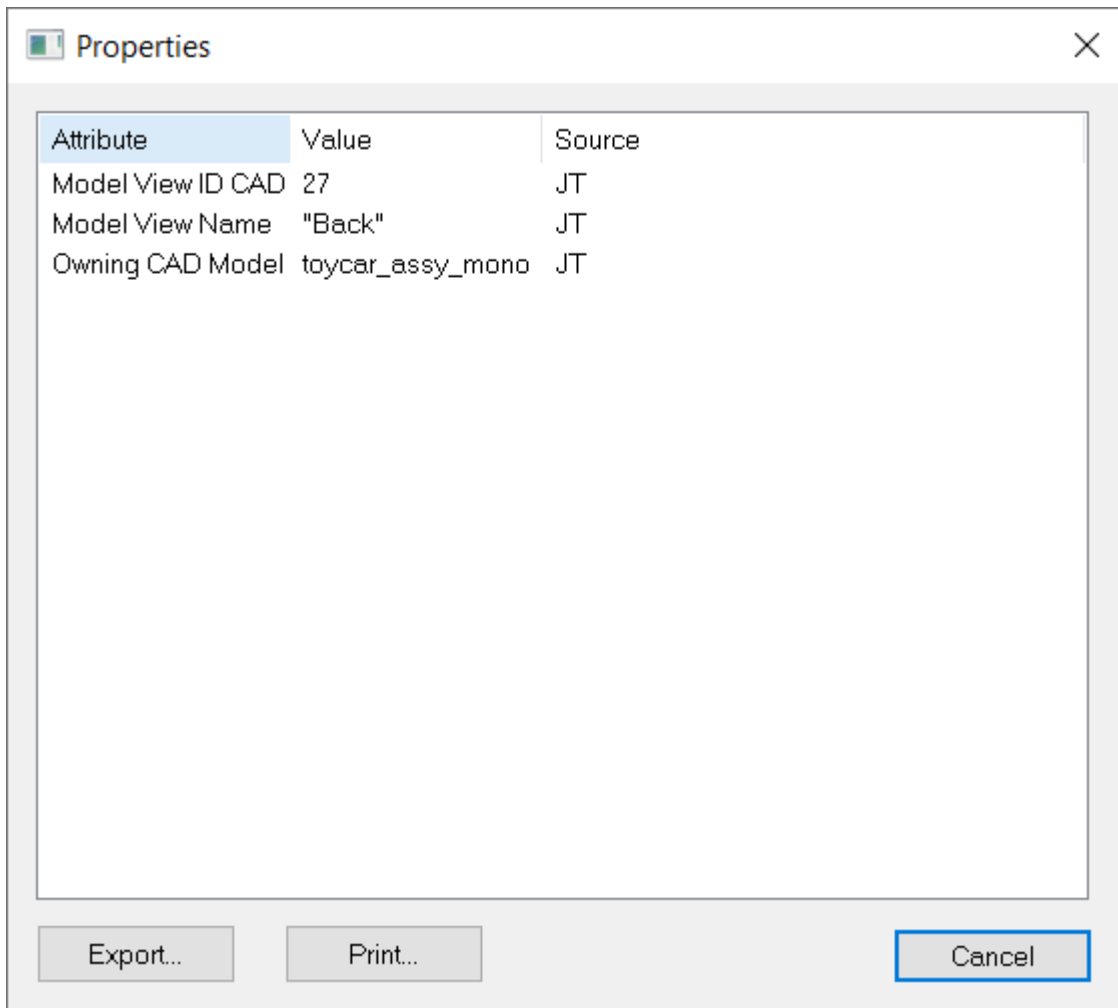
Sort model views

You can sort the list of model views that appears in the Model Views view in the Project Workspace window.

- In the Model Views view, right-click the blank area, choose **Sort**, and choose one of the following:
 - **Owning Part Ascending**: Sort the model views, by part, in ascending alphanumeric order.
 - **Owning Part Descending**: Sort the model views, by part, in descending alphanumeric order.
 - **Model View Part Ascending**: Sort all the model views in ascending alphanumeric order.
 - **Model View Part Descending**: Sort all the model views in descending alphanumeric order.
 - **None**: Model views are sorted in the order they are stored in the file.

Display model view properties

1. In the PMI view, right-click the model view for which you want to display properties, and choose **Properties**.



2. To save the properties as a .txt file, click **Export**.
3. To print the properties, click **Print**.
4. When you are done viewing the properties, close the **Properties** dialog box, or click **Cancel**.

Apply model views

Apply model views to update the 3D model in the Viewing window with the content of the model view. When you apply a model view, the product structure is loaded; the appropriate nodes are loaded from Teamcenter, if connected; and the model view at the selected node is displayed.

Apply models in the PMI view

1. Click the circle next to the model view you want to apply.

The circle appears red, indicating the model view is applied.

- To revert, either click the red circle next to it, or apply another model view.

The PMI label associated with the model view is no longer displayed in the Viewing window.

Working with PMI effectivity

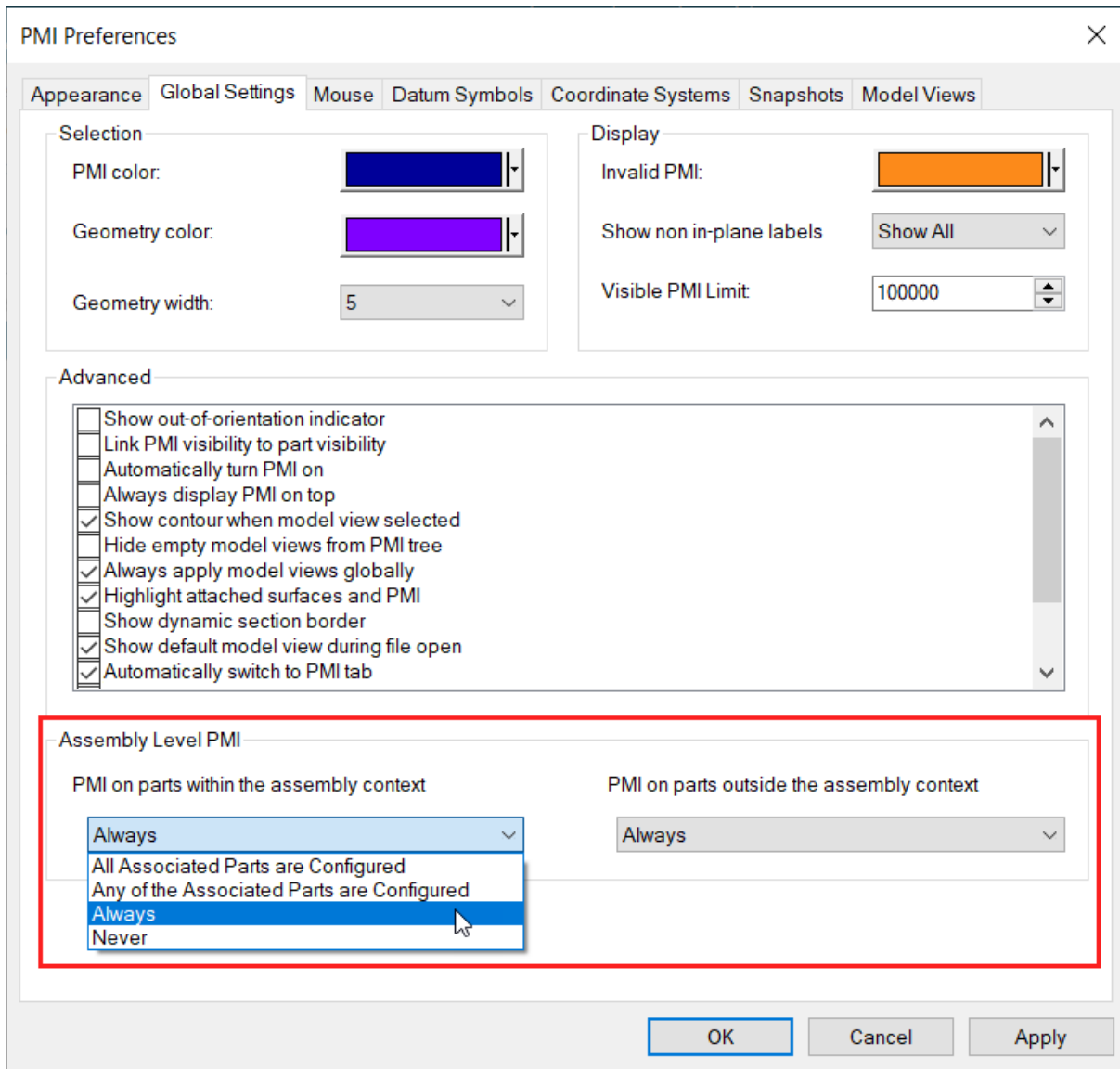
Understanding the workflow of PMI effectivity

If you are an NX and Teamcenter customer, you can configure and organize PMI based on effectivity by using the organizing structure generated by your company. For example, your assemblies and subassemblies might be organized by tail number. Based on the structure, you can create effectivity configurations in NX, then save those configurations to Teamcenter, and then send the saved dataset to a Teamcenter viewer from Teamcenter. Use the viewer to display and mark up the assembly.

Configure PMI effectivity preferences for Teamcenter lifecycle visualization

- In the Viewer, open a subassembly or assembly.
- Choose **View**→**Toolbars**, and then select **3D PMI**.
- In the **PMI Preferences** dialog box, select the **Global Settings** page.
- In the **Assembly Level PMI** section, choose **PMI on parts within the assembly context** or **PMI on parts outside the assembly context**, and then select an option from the list.

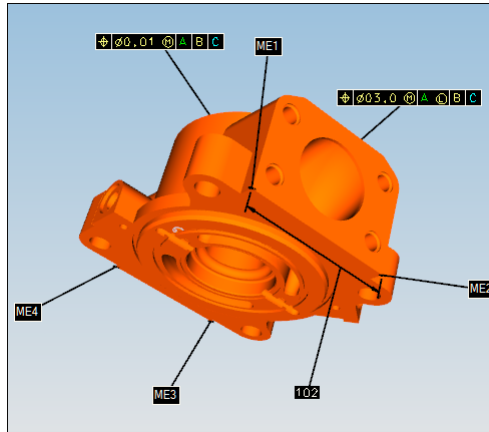
To	Select this option
Configure PMI as effective only when all referenced parts are effective.	All Associated Parts are Configured
Configure PMI as effective when any of its referenced parts are effective.	Any of the Associated Parts are Configured
Configure PMI to always display without checking for effectivity.	Always
Configure application so that PMI is not displayed.	Never



Working with PMI labels

Overview of working with PMI labels

You can choose what PMI to display in the Viewing window along with its appropriate label. Labels give you information about the specific PMI that is displayed and they can be attached to B-Rep entities of the PMI element. You may need to modify how the PMI label is displayed in the Viewing window to make it easier to view PMI elements and their labels.



Align PMI labels to an axis to make them easy to read

Use **Align View** to align the PMI label to an axis in the Viewing window so it is easy to read. Choose **Align View** from the following locations.

- In the Lifecycle Viewer, right-click a PMI label.
- In the PMI view, right-click an object with PMI data.
- In Model Views, right-click an object with PMI data.

Note:

If the Viewing window displays several coordinate system labels, drag and move the labels so you can align the view to one axis.

Center the PMI label in the Viewing window

Use **Seek** to move the PMI label to the center of the Viewing window. Choose **Seek** from the following locations.

- In the Lifecycle Viewer, right-click a PMI label.
- In the PMI view, right-click an object with PMI data.
- In Model Views, right-click an object with PMI data.

Hide a PMI label

Right-click a PMI label and choose **Hide**.

Move PMI labels forward and back

(When in flat-to-screen viewing mode and when labels are stacked on top of each other) Right-click a label and choose **Raise** to move the label toward you, or choose **Lower** to move the label away from you in the Viewing window.

Control the appearance and behavior of PMI entities

Set PMI appearance and behavior preferences

1. On the **PMI** toolbar, click **Preferences** .
2. In the **PMI Preferences** dialog box, change any of the following preferences:

On this page	Use this option	To
Appearance	Display Mode — In-Plane	Display PMI annotations in the plane of the <i>entity</i>, so they rotate with the model view. PMI data with parallel-to-screen CAD settings override the In-Plane functionality.
	Display Mode — Flat-To-Screen	Display PMI annotations in the plane of the <i>screen</i>, so they do not rotate with the model view. PMI data with parallel-to-screen CAD settings do not override the Flat-To-Screen functionality. Note: Borders can only be applied to PMI that are In-Plane.
	Foreground color	Change the foreground color of PMI labels.
	Background color	Change the background color of PMI labels.
	Background opacity — In-Plane	Increase or decrease the background opacity of in-plane PMI.
	Background opacity — Flat-To-Screen	Increase or decrease the background opacity of flat-to-screen PMI.
	Border	Show or hide a border around the PMI.
	Font size	Change the font size of PMI labels. Note: Changing the font size is only supported for PMI that are Flat-to-Screen.
	Foreground color	Change the color of the non-text portion of the PMI.

On this page	Use this option	To
	Width	Change the line thickness of the non-text portion of the PMI.
	Line style	Change the line style of the non-text portion of the PMI.
	Advanced	Set the appearance properties as default for all or selected PMI. Note: For JT files, when you set the preferences as default fallback using the Advanced option, the appearance properties defined in the PMI Preferences dialog will override the existing appearance properties, except if they have been already defined. Example: A PMI has a Foreground Color of green defined in the JT file and has nothing defined for Background Opacity. When you display the PMI, you see that the PMI text color is green (defined in the JT file) and a completely transparent background. Now, if you set the Foreground Color to white and the Background Opacity to 80% in the PMI Preferences dialog, then use the Advanced option to set these preferences as default, close the product and re-open the product, and display the PMI, you will see that the Foreground Color is green (because the property defined in the JT file overrides the default fallback that was set to white), but the Background Opacity remains at 80% (because this property wasn't defined in the JT file and was set to 80% as the default fallback value.)
Global Settings	PMI color	Select the highlight color for selected PMI.
	Geometry color	Set the color for the part geometry for selected PMI.
	Geometry width	Set the line width for the part geometry for selected PMI.
	Invalid PMI	Select the foreground color for PMI that is invalid.
	Show non in-plane labels	Choose how you want non in-plane labels displayed. Choose from the following: • Show All – Show all non in-plane labels. • Hide Welds – Hide the welds, but show all other non in-plane labels. • Hide All – Hide all non in-plane labels.
	Visible PMI Limit	Set the limit for the number of visible PMI objects displayed in the Viewing window. While there is no limit to the number of displayed objects, be aware that the greater the number of displayed objects, the slower the performance.

On this page	Use this option	To
	Link PMI visibility to part visibility	Display PMI labels attached to visible parts and hide PMI labels attached to parts that are turned off.
	Always display PMI on top	Make PMI labels always appear in the front of the model.
	Hide empty model views from PMI Tree	Hide empty model views from the PMI tree. An empty model view is any model view that does not contain visible PMI.
	Highlight attached surfaces and PMI	Highlight both attached surfaces and attached PMI. Note: CAD programs can create attached and associated surfaces. By default, both surfaces are highlighted.
	Show default model view during file open	Apply default model views to 3D models when you open them. If this option is not selected or the 3D model does not have default model view, no model view is applied when you open the 3D model. Note: If this option is selected, and if PMI part geometry is included in the default model view, the PMI will display when the model is open. To turn off the PMI, choose Manufacturing tab→PMI group→Enable PMI .
	Show out-of-orientation indicator	Display a red text markup in the Viewing window whenever a model view is altered from its original orientation state.
	Automatically turn PMI on	Display PMI automatically when parts and assemblies are loaded. Note: This option is active only when Link PMI visibility to part visibility is selected.
	Show Contour when Model View Selected	Display a line around the model where a section cut will occur.
	Always apply model views globally	Hide all parts outside the scope of the model view before a model view is applied. The scope of a model view is all parts that are descendants of the node that the model view is on, including the model view node itself. Note: If you clear this option, the JT file determines the part visibility.
	Show dynamic section border	Show or hide section lines and section borders.

On this page	Use this option	To
	Automatically switch to PMI tab	Automatically switches to the PMI page when Show PMI or Toggle in PMI Tree is selected from the context menu.
	Automatically reorient PMI	Automatically reorients the PMI annotations so that they remain readable as the view is rotated.
	Automatically populate PMI tree	Automatically populates the PMI tree on the PMI page.
	PMI on parts within the assembly context	From the list, you can select from the following options: • All Associated Parts are Configured – check the effectivity of each part the PMI references. If all parts are effective, the PMI is effective. • Any of the Associated Parts are Configured – check the effectivity of each part the PMI references. If any of the parts are effective, the PMI is effective. • Always – do not check effectivity; display the PMI. • Never – do not check effectivity; discard the PMI.
	PMI on parts outside the assembly context	From the list, you can select from the following options: • All Associated Parts are Configured – check the effectivity of each part the PMI references. If all parts are effective, the PMI is effective. • Any of the Associated Parts are Configured – check the effectivity of each part the PMI references. If any of the parts are effective, the PMI is effective. • Always – do not check effectivity; display the PMI. • Never – do not check effectivity; discard the PMI.
Mouse	Double Click Behavior	Turn on one of the following double-click behaviors: • Select Align View to PMI to align the 3D view to a PMI entity. • Select Display Properties to launch property data for single PMI entities. • Select Follow Hyperlink to open an active (underlined) link in a browser.
Datum Symbols	Datum Color (for the appropriate datum symbol)	Change the color of datum symbols.
Coordinate Systems	Trihedron Size	Change the overall size of the trihedron. Drag the slider to the left to decrease the size of the trihedron, or to the right to increase the size.
	Axis width	Set the line width for the coordinate system.

On this page	Use this option	To
	Axis style	Set the line style for the coordinate system.
	Show axis planes	View the axis plane of the coordinate system along with the axis lines.
	Show coordinate system using X-Y-Z instead of I-J-K	Set the axis label for the coordinate system PMI type from I-J-K to X-Y-Z.
Snapshots	Restore saved PMI on snapshot launch	Set the system to retrieve any saved PMI when the snapshot is opened.
	Capture visible PMI in the snapshot	Automatically save any visible PMI to the snapshot.
	Caption for PMI that cannot be restored	Apply a label to any PMI missing from the snapshot.
	Display reconciliation information when reattachment fails	Display reconciliation information when PMI information cannot be reattached to the geometry.
Model Views	Generate missing thumbnails on Find	Generate model view thumbnails for model views that do not already have associated thumbnails, when new model views are added to the Model Views view using Find. Model views will load more slowly, so this may not be recommended for models with a large number of model views.
	Update images on selected part change	When the Scope is Selected Parts , automatically find and show model views on selected parts, when the selection changes.
	Show tree view by default	Show the model views in the tree view, instead of the thumbnails view, when model views are loaded in the Model Views view.
	Show model views by default on load	Find and show all the model view thumbnails (actual, generated, or placeholder) when the Model Views view is opened.
	Show owning part in thumbnail names (Gallery)	Show owning parts on thumbnails in the Model Views view.
	Show owning part in thumbnail names, when available (Palette)	Show owning parts on thumbnails in the Model View Palette.
	Reconcile Palette Model Views on Palette Load	Reconcile model view palettes when models are loaded. When this option is not selected, model view palettes are only reconciled when you choose the option to reconcile them.
	Show Reconcile Configuration Mismatch warning	Display a warning message when a loaded structure configuration and palette structure configuration do not match.
	Generate high def images if not present	Generate high definition images of model views when thumbnails are generated or not present. Specify any of these options:

On this page	Use this option	To
		• Width and Height of the images, in pixels. • File Format: • jpg From the Quality list, select the quality level of an exported JPEG by setting the value from 1–100. The default is 98. When you increase the quality of the image by increasing this value, you also increase the size of the file. You can often decrease the file size and maintain acceptable results by lowering the value. • tif From the Compression list, specify a compression option for the TIFF file. • png • To adjust performance settings to provide the higher quality images, select Use high image quality for high def and thumbnail images. Performance settings are reset to previous settings after images are generated. • To set a memory limit for generating images, enter a value for the option High Def image memory limit.

Note:

For **Global Settings**, the following entities can only be displayed **Flat-To-Screen** regardless of the display type preference setting:

- Certification Points, Clinches, Coordinate Systems, Datum Locators, Datum Points, Face Attributes, Measurement Locators, Measurement Points, Structural Adhesives, and Welds.

3. (Optional) If you are displaying PMI in **In-Plane** mode and the text is not oriented correctly, on the **PMI** toolbar click **Reorient Text** .
4. Click **Apply** to apply the changes to your model without closing the dialog box, or click **OK** to apply the changes and close the dialog box.

Overview of temporarily changing the appearance of PMI entities

You can change PMI visibility states to modify the appearance of PMI entities in the Viewing window. Appearance changes are not persisted when you close the session.

You can:

- Change the appearance properties of one or more entities.
- Change the appearance properties of the entire model.

Temporarily change appearance of selected PMI entities

1. Open the **Appearance** dialog box in one of these ways:
 - In the Viewing window, select one or more entities, right-click and choose **Appearance**.
 - In the **PMI** view, choose a node, right-click and choose **Appearance**.

Note:

If you select a root or subassembly in the **PMI** view, all PMI that are children of the root or subassembly are affected.

2. Choose from the following options:

Use this option	To
Display Mode — In-Plane	Display PMI annotations in the plane of the entity, so they rotate with the model view. PMI data with parallel-to-screen CAD settings override the In-Plane functionality.
Display Mode — Flat-To-Screen	Display PMI annotations in the plane of the screen, so they do not rotate with the model view. PMI data with parallel-to-screen CAD settings do not override the Flat-To-Screen functionality.
Foreground color	Change the foreground color of PMI labels.
Background color	Change the background color of PMI labels.
Background opacity — In-Plane	Change the background transparency of in-plane PMI labels.
Background opacity — Flat-To-Screen	Change the background transparency of flat-to-screen PMI labels.
Border (In-Plane only)	Display a border around the PMI label.
Font size (Flat-to-Screen only)	Change the font size of PMI labels.
Foreground color	Change the color of the non-text portion of the PMI.

Use this option	To
Width	Change the line thickness of the non-text portion of the PMI.
Line style	Change the line style of the non-text portion of the PMI.

- Click **Apply** to apply the changes to the selected entities.
- Click **OK**.

Temporarily change PMI appearance for the entire model

- In the **PMI** view, select the top node (in the example file the top node is *PMI*), and then right-click and choose **Appearance**.
- In the **Appearance** dialog, choose from the following options:

Use this option	To
Display Mode - In-Plane	Display PMI annotations in the plane of the entity, so they rotate with the model view. PMI data with parallel-to-screen CAD settings override the In-Plane functionality.
Display Mode - Flat-To-Screen	Display PMI annotations in the plane of the screen, so they do not rotate with the model view. PMI data with parallel-to-screen CAD settings do not override the Flat-To-Screen functionality.
Foreground color	Change the foreground color of PMI labels.
Background color	Change the background color of PMI labels.
Background opacity - In-Plane	Change the background transparency of in-plane PMI labels.
Background opacity - Flat-To-Screen	Change the background transparency of flat-to-screen PMI labels.
Border (In-Plane only)	Display a border around the PMI label.
Font size (Flat-to-Screen only)	Change the font size of PMI labels.
Foreground color	Change the color of the non-text portion of the PMI.
Width	Change the line thickness of the non-text portion of the PMI.
Line style	Change the line style of the non-text portion of the PMI.

3. Click **Apply** to apply the changes to the entire model.
4. Click **OK**.

Generate reports about measurement points and spot welds

1. On the **PMI** toolbar, click **Multiple Entity Mode**.
2. In the Viewing window, select the parts to include in the report.

PMI associated with the selected parts appears.

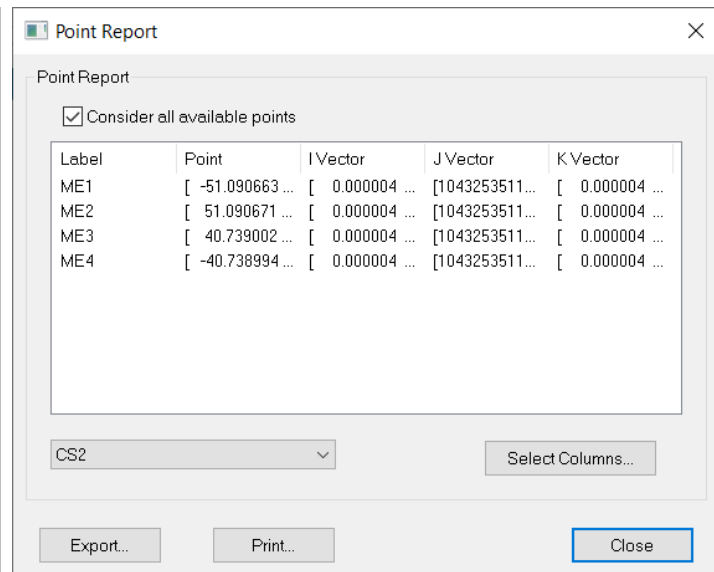
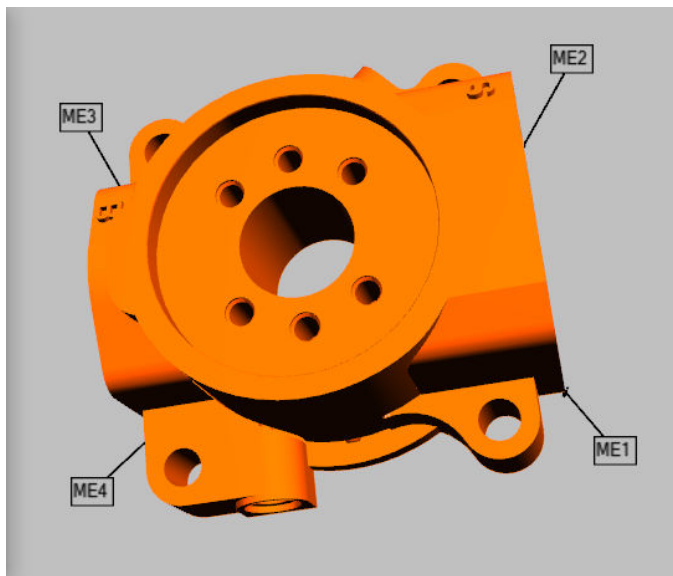
3. (Optional) To hide all other PMI, **configure the filter** to display one or more of the following:

- **Measurement Points**
- **Spot Welds**

4. Do one of the following:

To	Do this
Include only selected measurement points and spot welds.	a. In the Viewing window or PMI tree, select the measurement and spot weld points to include. (Press Ctrl to select multiple points.) b. Choose PMI→Point Report.
Include all measurement points and spot welds.	a. Choose PMI→Point Report. b. In the Point Report dialog box, select the Consider all available points check box.

5. The **Point Report** dialog box displays the report.



6. (Optional) Choose **Select Columns** to add or remove columns.
7. (Optional) In the section to the left of **Select Columns**, select the coordinate system to use.
8. (Optional) Click **Export** to save the report to a text file.
9. (Optional) Click **Print** to print the report.
10. Click **Close**.

Viewing CAE results

Viewing CAE results overview

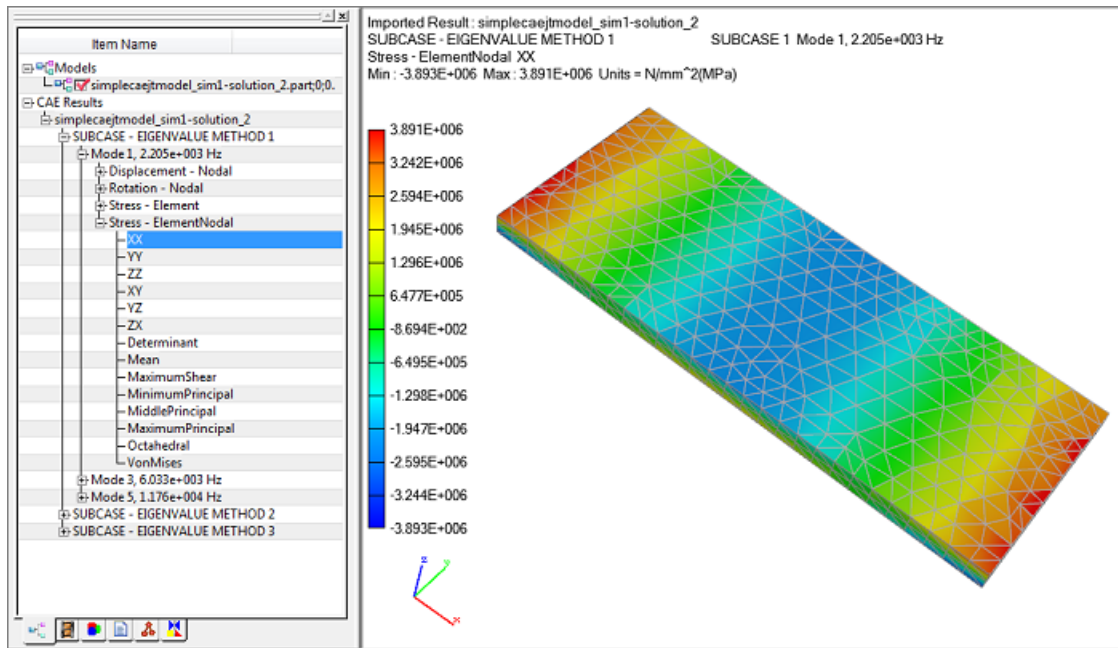
You can view Computer Aided Engineering (CAE) Finite Element Analysis (FEA) results exported as a .jt file from a supported CAE application.

Currently supported CAE applications include NX, I-deas, and FEMAP.

Note:

- In Professional and Mockup, you can view, probe, and annotate CAE results in v10 JT (or later) files created in NX 10 (or later).
- You have limited viewing capabilities with CAE results in v9.x JT (or earlier) files created in NX, I-deas, or FEMAP.

Viewing CAE results in v10 JT files



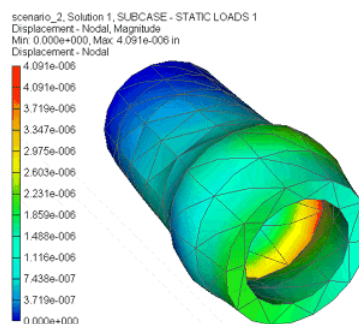
With CAE results in v10 JT (or later) files created in NX 10 (or later), you can:

- In Professional and Mockup, view multiple results in a single .jt file.
- In Professional and Mockup, probe the CAE results on the CAE model.
- In Professional and Mockup, create CAE annotations and persist those annotations into session or PLM XML files.

Note:

A graphics card supporting OpenGL 3.3 or greater is required to visualize CAE data stored in JT 10 files. There is no graphic requirement for earlier versions of .jt files.

Viewing CAE results in v9.x JT files



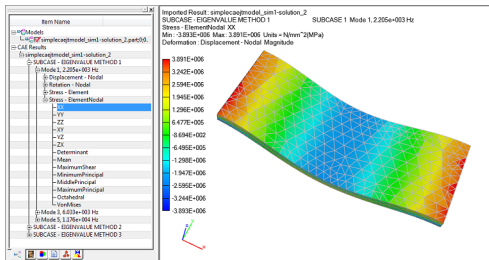
With CAE results in v9.x JT (or earlier) files created in NX, I-deas, or FEMAP, you can:

- View results from a single analysis.
- Show and hide markups created in the CAE application.

Display CAE information

1. Load a .jt file with CAE results.

The .jt file appears in the 3D window with the first CAE result component displayed. The color bar and legend update for the current selection. The information for the color bar and legend are collected from the .jt file. The CAE result tree appears in the Assembly page of the Project Workspace.



2. Expand the CAE results in the Assembly page of the Project Workspace, and double-click CAE results to view.

Note:

For Base and Standard service levels, you can view only the first result.

Set CAE annotation preferences and properties

You can change the appearance of CAE annotations.

1. To change the default appearance of all annotations, on the **3D CAE Viewing** toolbar, click **Preferences** .

-or-

To change the properties of an individual annotation, right-click the annotation and select **Properties**.

2. Adjust any of the following settings

On this tab	Use this option	To do this
Line	Line style	Change the leader line style of CAE annotations.
	Line color	Change the color of CAE annotation leader lines.
	Thickness	Specify the thickness of CAE annotation leader lines. Type a larger number to draw a thicker line, or a smaller number to draw a thinner line.
	Line Ends Style	Change the style of the ends of CAE annotation leader lines. a. Choose a style in the Begin column for the beginning of your lines (where the annotation begins). b. Choose a style in the End column for the end of your lines (where the leader line attaches to the model).
	Line Ends Size	Change the size of the leader line style selected in the previous step. a. Choose a size in the Begin column for the beginning leader line style. b. Choose a style in the End column for the end leader line style.
Edge	Edge style	Change the border style for CAE annotations displayed with a filled background.
	Edge color	Change the border color of CAE annotations displayed with a filled background.
	Thickness	Change the border thickness of CAE annotations displayed with a filled background. Type a larger number to draw a thicker edge, or a smaller number to draw a thinner edge.
Fill	Fill enabled	Enable or disable a filled background for CAE annotations.
	Fill style	Change the filled background style for CAE annotation.
	Fill color	Change the filled background color for CAE annotations.
	Transparency	Change the filled background transparency for CAE annotations. Drag the slider toward Transparent to make objects behind the background more visible. Drag the slider toward Opaque to make the background more visible.

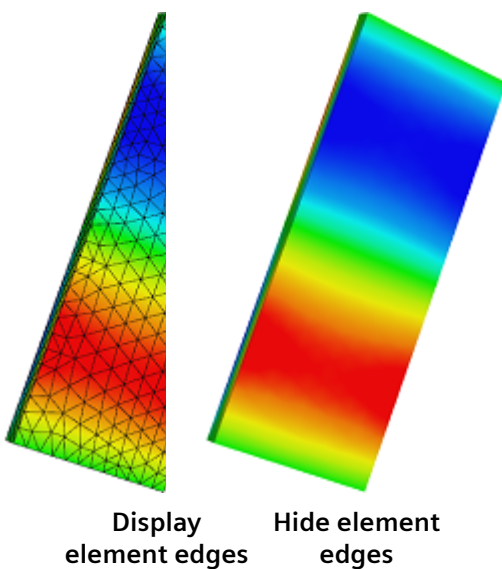
On this tab	Use this option	To do this
Font	Font	Change the font used for text in CAE annotations.
	Font Style	Change the font style of text in CAE annotations. Note: Not all font styles may be available.
	Size	Change the font size of text in CAE annotations.
	Effects	Add or remove underline or strikeout text effects in CAE annotations.
	Color	Change the font color for text in CAE annotations.

- (Optional) To preview individual annotation properties, click **Apply**.
- Click **OK**.

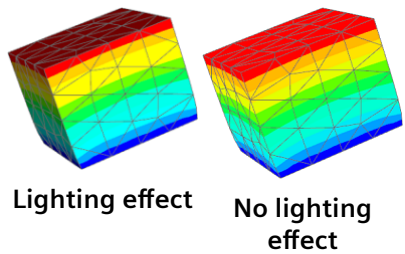
Modify the model display

Display element edges and lighting effects

- On the **3D CAE Viewing** toolbar, click **Display Options** .
- In the **Display Options** dialog box under the **Other** section, set **Display Element Edge** to display or hide element edges.



- Remove the light location effect so no variance exists between the color bar and the model colors.



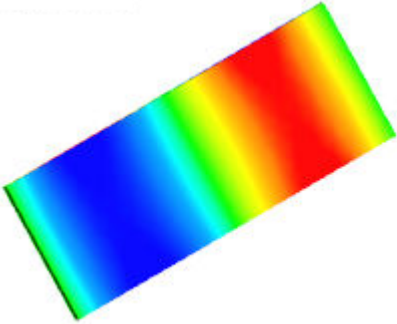
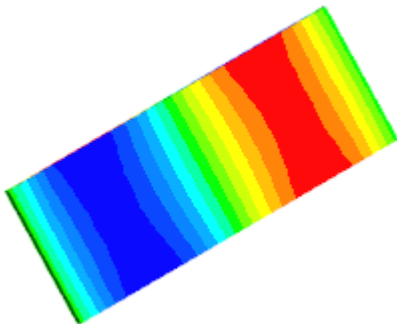
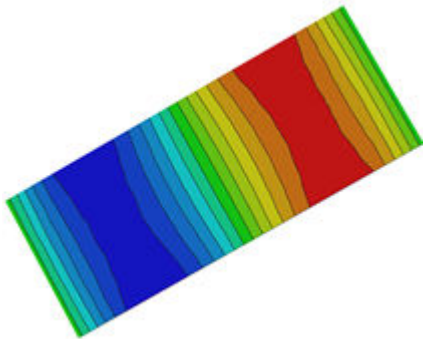
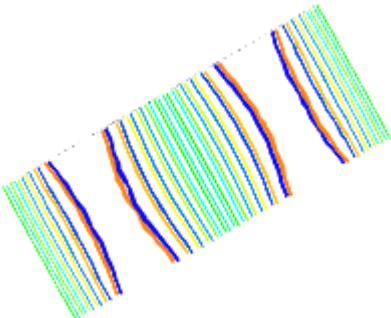
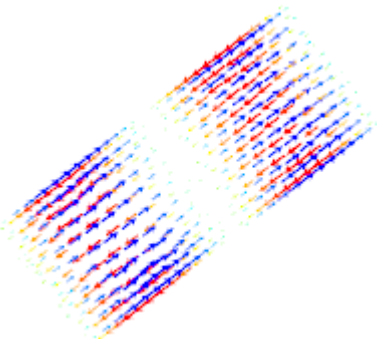
- (Optional) To preview individual properties, click **Apply**.
- Click **OK**.

Set the color display

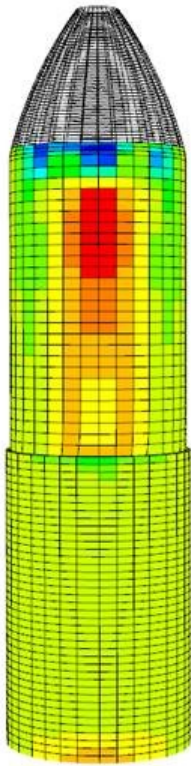
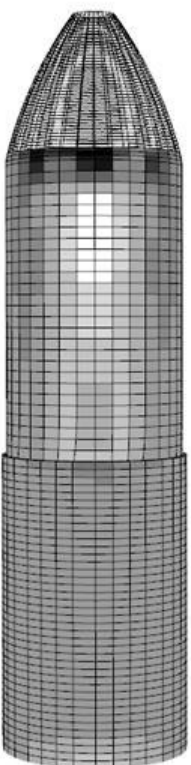
Set the color display for selected CAE results.

- On the **3D CAE Viewing** toolbar, click **Display Options** .
- In the **Display Options** dialog box, select **Color Display** to display the result colors.
- Click **Configure**.
- Define the color options.

In this section	Use this option	To
Result Selection	Subcase Step Result Component	Determine which CAE result displays in the Viewing window. The values correspond to the options available in the assembly tree. The only results that appear in the Result list are those to which the selected Color Mode can be applied.
Legend and Display	Color Mode	Define the style in which CAE results are displayed on both the legend and the model. Smooth , Banded , and Banded Isoline create a colored overlay on the model, while Isoline displays only contour lines. Arrow Mode displays arrows reflecting CAE values.

In this section	Use this option	To
		  Smooth Band   Banded Isoline Isoline  Arrows
	Legend Extremes	Specify how the color bar scales the colors across the range of values. Result Scales the color bar across the complete range of values for the selected result for the entire model. Specified Scales the color across a specified range of values. If you select Specified, enter the Minimum and Maximum values.
	Minimum	Available when Legend Extremes is set to Specified .

In this section	Use this option	To
		Set the minimum values for the color bar. This value determines the color range for both the legend and the model.
	Maximum	Available when Legend Extremes is set to Specified . Set the maximum values for the color bar. This value determines the color range for both the legend and the model.
	Value Display	Determine how numbers are displayed on the legend. Standard Numbers display as regular decimals, for example, 0.123. Scientific Notation Numbers display as exponent notation, for example, 1.23E-1.
	Legend	Specify the spacing values between the numbers on the legend. Equally Spaced Assures the distance between any two numbers on the legend is equal. The definition of "equally spaced" is different depending on whether the Scale option is set to Linear or Logarithmic . Round Off Maintains close to equal spacing between legend values, but rounds numbers to set intervals, which may make the legend values not precisely equally spaced. Level Increment Maintains a custom distance between all legend values, which is determined by the value specified in the Increment Per Level field.
	Scale	Specify the scale applied to the levels in the color bar. Choose from Automatic , Linear , and Logarithmic .
	Max Levels	Specify the number of color levels for the color bar. Choose a value between 1 and 50.
	Increment Per Level	Available when Legend is set to Level Increment . Specify the increment amount.
Color Control	Spectrum	Specify the color spectrum assigned to the results values.

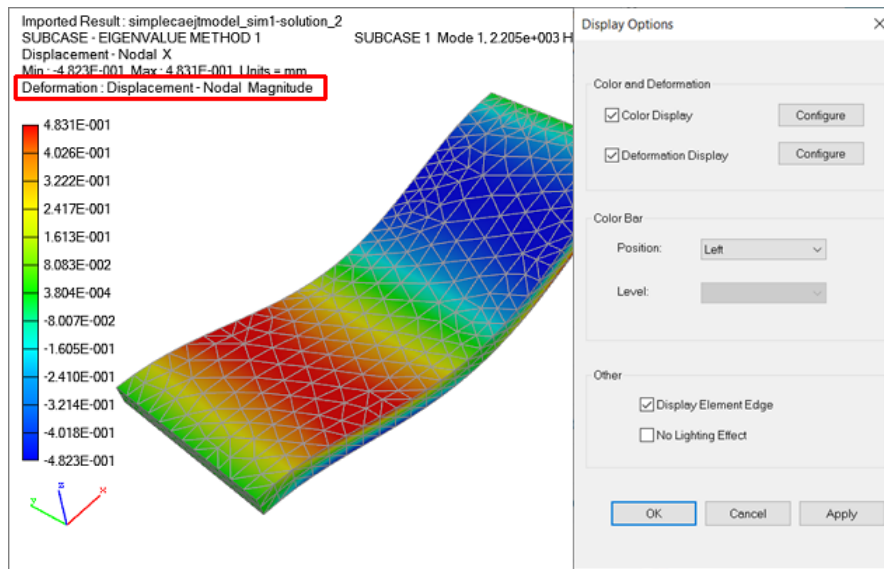
In this section	Use this option	To
		 Structural Thermal  Greyscale Stoplight

In this section	Use this option	To
	Invert Spectrum	Invert the selected color spectrum. For example, if you invert a structural spectrum, the highest values are blue and the lowest values are red.
Overflow/Underflow	Overflow Underflow	Specify the color used for values outside the specified range. Click the color well to modify the specified color. None Shaded Translucent Clipped
Other	Arrow	Available when Color Mode is set to Arrows . Determine the size of the arrows as a percentage of the model size. The default is 5%, meaning that the length of the arrow is set to be 5% of the model size, defined as the diagonal length of the part bounding box, at the location where the result magnitude is greatest. The length of the arrow at other locations is scaled based on the result magnitude.

- (Optional) To preview individual properties, click **Apply**.
- Click **OK**.

Set the deformation display

Set the display of deformations for selected results. When you turn on the display of deformations, the deformation information appears in the legend at the top of the Viewing window. The display of deformation is off by default.



1. On the **3D CAE Viewing** toolbar, click **Display Options** .
2. In the **Display Options** dialog box, select **Deformation Display** to display the CAE deformation results.
3. Click **Configure**.
4. In the **Results Selection** preferences, select the result set to which you want to apply the deformations.

The only results that appear are those to which deformations can be applied.

Note:

Subcase and **Step** are **set** to the same values as **Subcase** and **Step** in the **Color Display** dialog box.

5. Specify **Scale** for the magnitude of the deformation as a percentage of the model.

Note:

The default value for **Scale** is always 10% of the model.

6. (Optional) To preview individual properties, click **Apply**.
7. Click **OK**.

Set the location of the color bar

Specify the location of the color bar. The color bar, the text of its legend, and the range of colors are stored in the .jt file.

1. On the **3D CAE Viewing** toolbar, click **Display Options** .
2. In the **Display Options** dialog box under the **Color Bar** section, set **Position** to the location of the color bar.
3. Set **Level** to display the color bar sections as:
 - **Smooth**
 - **Band**
 - **Scalar**
 - **Discrete**
4. (Optional) To preview individual properties, click **Apply**.
5. Click **OK**.

Probe CAE results

Use **Identify** to probe and display CAE information stored in a .jt file. Hover over a node or element to display the CAE information, or select the node or element to create a temporary anchored markup. You can display entity IDs or results values on the probed results. You can probe nodal, elemental, and element nodal results.

Note:

- Visualize the results with or without deformations. Lifecycle Visualization always displays the same results for the queried entity regardless of whether the deformation display is on or off.
- Only exterior elements can be queried. To help locate elements in the viewport, **make exterior element edges** visible before probing.

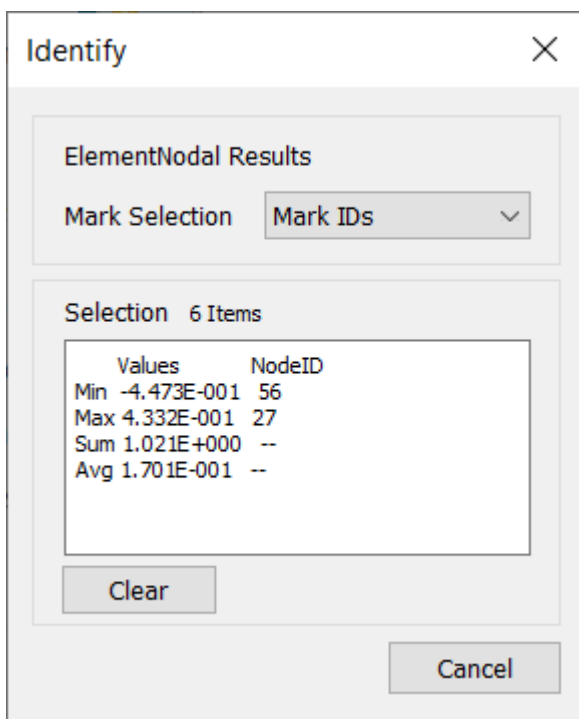
Tip:

To capture the markups permanently, **create** annotations.

1. (Optional) **Turn on edges** to help locate nodes and elements.

2. (Optional) **Turn on** deformations.
3. On the **3D CAE Viewing** toolbar, click **Identify** .
4. Select whether you want to display temporary markups when probing:
 - **Mark IDs** – Display the entity ID.
 - **Mark Result Values** – Display the result value.
 - **No Marks** – Displays an indication.
5. Hover over an entity to view the results values at that location, or select the entity to display a markup.

When you select entities, the **Identify** dialog box displays a summary of the minimum, maximum, sum, and average results values of the selected entities.



6. (Optional) Click **Clear** to clear the selection of entities and the summary report in the **Identify** dialog box.
7. Click **Cancel** to exit probing.

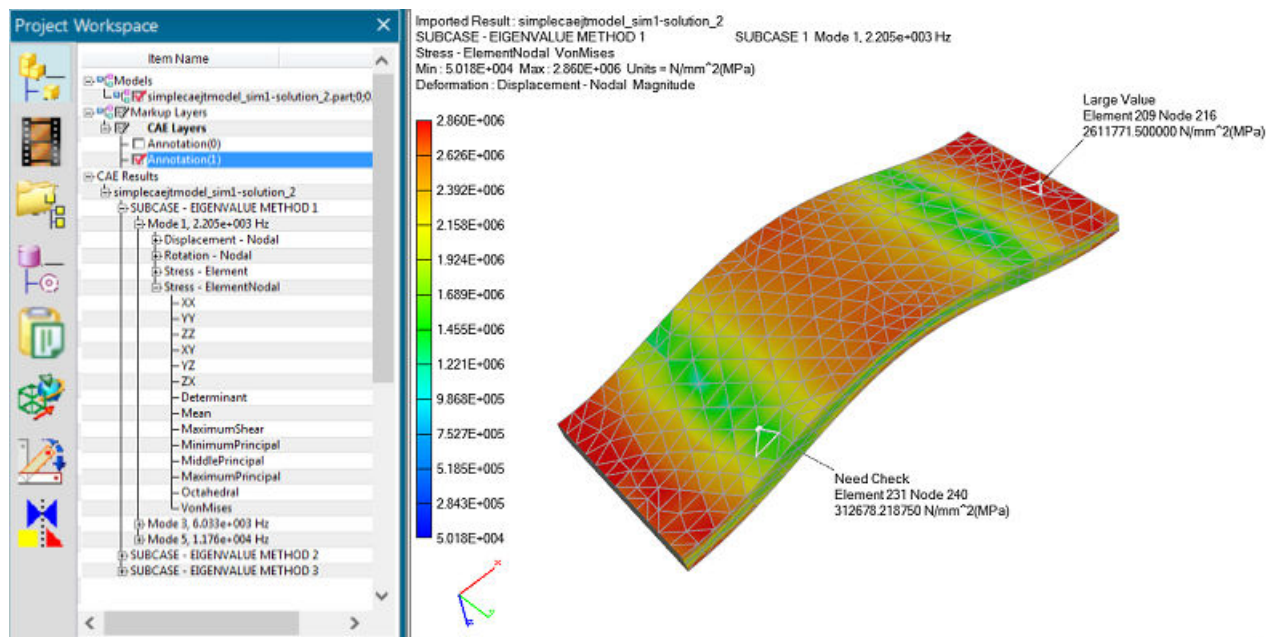
Lifecycle Visualization clears any temporary markups.

Annotate CAE Results

About annotating CAE results

Create user-defined annotations at selected nodes or elements to better communicate the CAE results. The annotations automatically display the CAE information of the selected entities, including result values, entity labels, and user-defined text.

The annotations are displayed in the Assembly page of the Project Workspace window. Save the annotations into a session or PLM XML file, and restore them when the file is loaded into Lifecycle Visualization.



Create CAE annotations

1. On the **3D CAE Viewing** toolbar, click **Annotations** .
2. Enter a title for the annotation.
3. In the **Selection** preferences, select **Mark IDs** to display the entity IDs of the results.
4. In the **Content** section, set the content of the annotation.

For this option	Do this
Value	Select one of the following: • Result and Entity ID

For this option	Do this
	• Result • Entity ID • None
User Text	Enter text to appear in the annotation to help further communicate the results.

5. In the **Display** preferences, set how the annotation is displayed.

For this option	Do this
Annotation type	Select one of the following: • Box with Leader Line • Box
Text Alignment	Select one of the following: • Left • Right

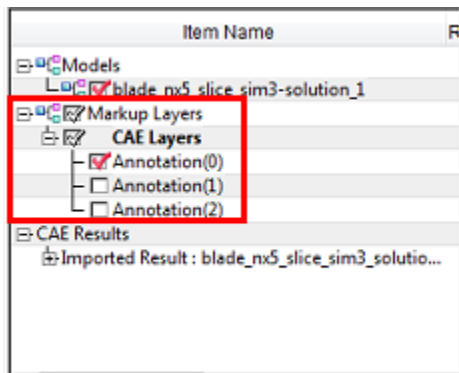
6. To **set** CAE preferences and properties, click **Change Annotation Color and Text Properties**.
7. In the Viewing window, select the locations at which to display the annotations.
8. (Optional) Click **Clear Highlight** to clear the highlighting of the selections, or click **Clear Selection** to clear the selections in the current annotation configuration.
9. (Optional) Click **Apply** to preview the annotations.
10. Click **OK**.

The annotations appear in the Viewing window. They also appear in the Assembly page of the Project Workspace window.

Apply existing CAE annotations

You can turn on or off the associated annotations on the CAE result display.

In the CAE Layer of the Assembly page of the Project Workspace window, select an annotation.



Capture CAE results with annotations in snapshots

You can capture CAE results with annotations in 3D snapshots.

1. On the **Snapshots** toolbar, click **Preferences** .

-or-

In the **Snapshots** view, right-click and choose **Preferences**.

2. Select the **CAE** check box.
3. Click **OK**.
4. Display the CAE information you want to capture in the snapshot.
5. On the **Snapshots** toolbar, click **Add** .

-or-

In the **Snapshots** view, right-click and choose **Add**.

Save annotations and snapshots

If you create annotations and capture CAE result display in snapshots and want them to be available the next time you work with the model, save your work to a session file or a .plmxml file.

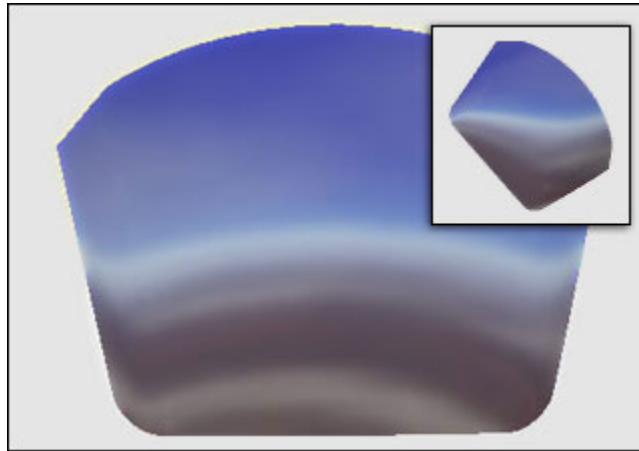
Do one of the following:

To	Do this
Save annotations and snapshots to a session file	Note: Annotations and snapshots automatically save when you save to a session file. 1. Open a model and capture one or more annotations and snapshots. 2. Choose File→Save Session.
Save annotations and snapshots to a .plmxml file	Note: You can choose to include annotations and snapshots when you save your work to a .plmxml file. 1. Choose File→Preferences→PLM XML. 2. On the Save page of the PLM XML Preferences dialog box, select the Save extended 3D content into PLM XML check box, and then click OK. 3. Choose File→Save As. 4. Type a file name and click OK.

Performing surface analysis

Perform reflection analysis

Perform reflection analysis to visualize how reflections distort across surface variations on your parts.



1. Select or display only the geometry that you want to include in the surface analysis.
2. On the **3D Surface Analysis** toolbar, click **Reflection** .
3. In the **Reflection Analysis** section of the **Surface Analysis** dialog box, choose from any of the following predefined images, or browse for an image of your own:

Lines: Thin black lines in a horizontal, vertical, or intersecting pattern. Lines overlay the underlying surface material, allowing it to show through in the spaces between the lines.	
Stripes: Zebra stripes or color bars in vertical and horizontal directions. This effect hides the underlying surface material.	
Horizon: Images used to simulate reflected environments. This effect blends with the underlying surface material.	

The image is mapped to the specified parts in the Viewing window.

4. (Optional) Adjust the **Image Scale** slider.

Note:

Images used for reflection analysis are tiled. Move the slider to the left to zoom closer to the tiled image; move the slider to the right to zoom out.

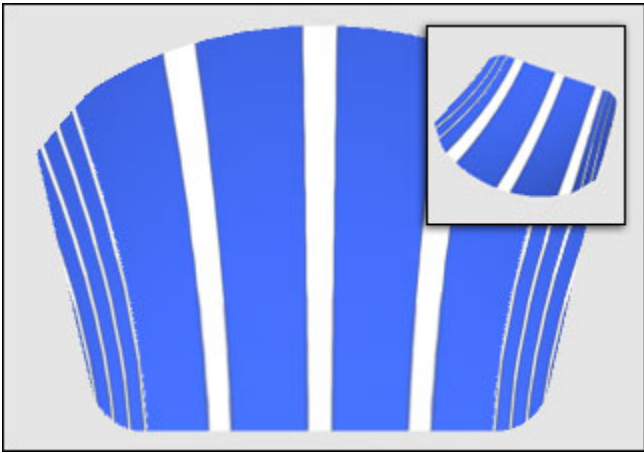
5. (Optional) Click **Parameters**.

The **Reflection Parameters** section appears. Adjust any of these settings:

Use this option	To do this
Line Thickness	Change line thickness for Lines images. You can specify the following degrees of thickness: • Thin • Medium • Thick
Stripe Edges	Change edge softness or hardness for Stripes images. You can choose from the following settings: • Hard • Soft

Apply flow lines to surfaces

Apply iso-parameter flow lines to your parts to identify their surface variations. Flow lines bend and distort across surface variations.



Note:

Flowline analysis requires parts with B-Rep data (JT B-Rep or XT B-Rep).

1. Select or display only the geometry that you want to include in the surface analysis.
2. On the **3D Surface Analysis** toolbar, click **Flow Lines** .

- In the **Flow Lines** section of the **Surface Analysis** dialog box, specify one of the following flow line variations:



The flow lines are mapped to the specified parts in the Viewing window.

- (Optional) Adjust the **Step Size** slider.

Note:

Move the slider to the left to zoom closer to the flow lines, making them look smaller and closer together; move the slider to the right to zoom out from the flow lines, making them look bigger and further apart.

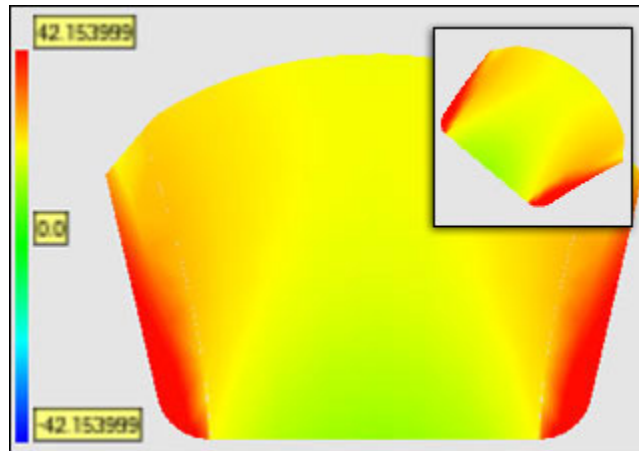
- (Optional) Click **Parameters**.

The **Flow Lines Parameters** section appears. Adjust any of these settings:

Use this option	To do this
Line Color	Change flow line colors. You can specify colors for both U (horizontal) and V (vertical) flow lines.
Line Width	Change flow line width. You can choose from the following settings: • Thin • Medium • Thick

Perform curvature analysis

Use curvature analysis to shade parts based on the curvature value of their surfaces. A color map that corresponds to curvature values is mapped to the specified parts in the Viewing window, hiding the underlying surface material.

**Note:**

Curvature analysis requires parts with B-Rep data (JT B-Rep or XT B-Rep).

1. Select or display only the geometry that you want to include in the surface analysis.
2. On the **3D Surface Analysis** toolbar, click **Curvature** .
3. In the **Surface Curvature** section of the **Surface Analysis Workspace** dialog box, select a method for visualizing the part curvature. Choices for visualizing surface curvature include **Minimum**, **Maximum**, **Average**, the three world coordinate axes, and **Gaussian**.

A color map that corresponds to curvature values is mapped to the specified parts in the Viewing window. By default, a color legend appears on the left side of the Viewing window.

Note:

- The color scale is a linear interpolation between five colors: red=maximum, yellow=median positive, green=zero, blue=median negative, violet=minimum.
- The values for the color scale are displayed as radii, using the model units (the unit of measurement specified before the model was translated).

4. (Optional) Adjust the **Radius Range** slider.
5. (Optional) Select **Specular Highlights**.
6. (Optional) Select **Absolute Values**.
7. (Optional) Click **Parameters**.

The **Curvature Parameters** section appears. Adjust any of these settings:

Use this option	To do this
Legend Placement	Change the location of the color legend in the Viewing window. You can specify the following settings: • Left • Right • Off
Transition	Change the transition between color values. You can choose from the following settings: • Soft • Hard
Specify Range	Change the range for the color map. This value is persisted between application sessions.

Apply random colors to surfaces

You can apply colors randomly to part surfaces, quickly identifying every surface of your parts. For each instance that you use **Color Surfaces Randomly**, a random color is applied to each surface of the parts specified in the Viewing window.



Note:

Random color analysis requires parts with B-Rep data (JT B-Rep or XT B-Rep).

1. Select or display only the geometry that you want to include in the surface analysis.

- On the **3D Surface Analysis** toolbar, click **Random Colors** .

-or-

On the **3D Surface Analysis** toolbar, click **Workspace**, and then, in the **Random Surface Colors** section of the **Surface Analysis** dialog box, click **Random Surface Colors**.

Random colors are applied to the surfaces of the parts specified in the Viewing window.

- (Optional) In the **Random Surface Colors** section of the **Surface Analysis** Dialog box, click **Restore**.

The part surfaces are returned to their initial colors.

Manage surface analysis settings and effects

Display a combination of surface analysis effects, replace the working set of parts, or show, hide, and adjust the tessellation quality of the effects.

- On the **3D Surface Analysis** toolbar, click **Workspace** .
- In the **Workspace** section of the **Surface Analysis Workspace** dialog box, do any of the following:

To	Do this
Display a combination of surface analysis effects.	- On the 3D Surface Analysis toolbar, click Multi-Effect . -or- - In the Management section of the Surface Analysis Workspace dialog box, click Multi-Effect Mode . - Combine surface analysis effects in any of these ways: - Curvature analysis and reflection analysis. - Reflection analysis and flowline analysis. - Random surface colors can be applied to parts in combination with any of the surface analysis effects.

To	Do this
	Note: • Curvature analysis effects cannot be combined with flowline analysis effects. • Combining curvature analysis and reflection analysis affects the display of curvature-based colors on parts. This combination works best with images of black lines.
Replace the working set of parts.	On the 3D Surface Analysis toolbar, click Set Working Set . -or- In the Management section of the Surface Analysis Workspace dialog box, click Set Working Set . Note: The working set of parts is created automatically when you perform the first surface analysis operation of a session. With an effect active and a working set defined, you can select or display a different set of parts, and click Set Working Set to replace the working set. If you have applied an effect and then disabled it, the Set Working Set button remains active, indicating that a working set is present even if no effects are enabled. If you click Set Working Set while no effects are enabled, the button resets, and becomes inactive.
Show or hide surface analysis effects.	On the 3D Surface Analysis toolbar, click Hide Effects . -or- In the Management section of the Surface Analysis Workspace dialog box, click Hide Effects .
Adjust the tessellation quality.	In the Management section of the Surface Analysis Workspace dialog box, for Tessellation Quality, choose one of the following: • Standard • Better • Best

To	Do this
	Tip: The flow lines, curvature analysis, and random color options re-tessellate your data in order to generate their effects. The time required to display the effects depends upon the size and number of the parts involved in the analysis. To shorten the amount of time it takes to perform your analysis, select or display fewer parts, and/or choose a lower Tessellation Quality setting, such as Standard.

Specify surface analysis preferences

You can specify to apply surface analysis effects automatically, without having to interact with the **Surface Analysis** dialog box. You can also choose to display an error message if an effect cannot be applied.

1. On the **3D Surface Analysis** toolbar, click **Preferences** .
2. Select the **Use Last Applied Settings (No Dialog)** check box.

When you choose a surface analysis type, the last used settings are automatically applied to the working set of parts. The **Surface Analysis** dialog box does not appear.

3. Select the **Show Effect Failed dialog** check box.

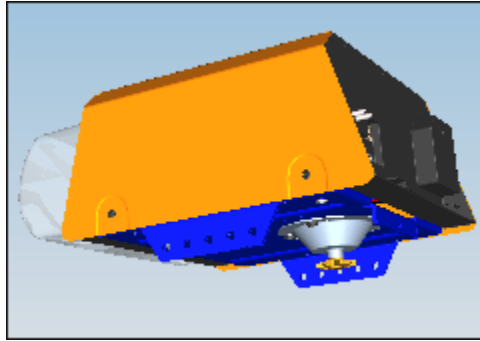
If a surface analysis effect cannot be applied to a part, an error message is displayed

Working with visual reports

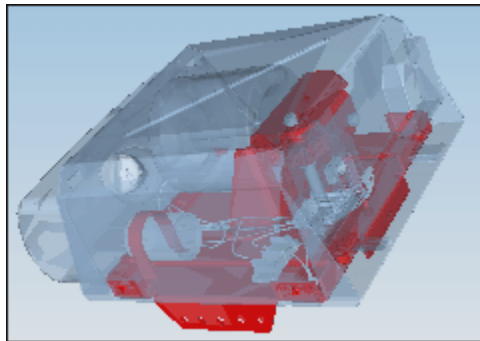
Overview of visual reports

Use visual reports to quickly display and analyze parts of an assembly in meaningful and well organized ways. You can create search queries using conditions and regular expression language. You create the scope of your searches by specifying part metadata attributes and values. Then you apply the report rules so you can see matched and unmatched parts in the Viewing window. The following example uses a garage door opener model.

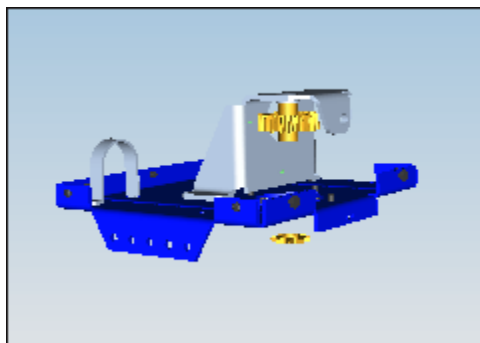
First, the original garage door opener is displayed.



Next, a visual report is created to display steel parts highlighted in red.



Finally, only steel parts are displayed in their original state.



While visual reports are most useful for displaying matched and unmatched parts of an assembly using unique color or display characteristics, you can also create reports using other conditions. For instance, you can create reports that show matched parts only (in their original display characteristics), and you can generate reports that hide matched or unmatched parts.

You might want to see the following types of information in a visual report:

- Parts that have changed over a specific period of time.
- Parts that you outsourced to a specific organization or group.

- Parts designed from a specific material (for example, steel).
- Parts that weigh more than a specified limit.
- Parts that are not yet complete and ready for release.

Visual reports are generated from attributes and values found in JT metadata (author, translation date, version, weight, material, and so forth) and from specific Teamcenter metadata.

Visual reports then use regular expression language to create robust search conditions. For example, you can generate a visual report that displays all steel parts in a color of your choice.

Using visual reports, you can:

- Retrieve and apply an existing visual report.
- Create visual reports using three supported reporting methods: a quick report, a report generated with a single report rule, and a report generated with multiple report rules.
- Review the results of visual reports by cycling through matched instances.
- Save visual reports as a .vpx file for later use.
- Save your results in snapshots and session files for future comparison.
- Edit any of the conditions and rules of the visual report.
- Reset the display of the model to its state prior to running the visual report.
- Specify visual report preferences, which include setting part colors, transparency levels, material options, display modes, and options for the report legend.

Note:

Using Visual Report requires an additional license and the feature is supported in stand-alone viewers at the Mockup and Professional service levels.

Working with visual reports using Teamcenter embedded viewers

Introduction

Teamcenter supports using visual reports in the embedded viewers, for example Structure Manager and CAE Manager. You configure visual reports to quickly display and analyze part and BOM line attributes. You can create search queries using conditions and regular expression language. To create the structure of your search, specify part and BOM line attributes and their values. Then **Apply** the report rules so you can see matched and unmatched parts in the viewing window. Optionally, you can then save your visual

report to the local file system, to a server, or to the Teamcenter database. For Teamcenter, you can also use Classification to specify how you want to sort and save visual reports for later use.

Access visual report tools in embedded viewers

You access the visual report tools for all active and supported Teamcenter embedded viewers by opening data in the viewer and then right-click anywhere in an existing toolbar, for example 3D Navigation. From the list of toolbars, select the **3D Visual Report** toolbar.

Use the following information as an introduction to visual report tools:



	Use this option	To do this
	Visual Report	Activate Visual Report tools.
	Apply Report	Open the Apply Report dialog box. Use the Apply Report dialog box to edit and load existing visual reports.
	Quick Report	Open the Quick Report dialog box. Use the Quick Report dialog box to quickly identify the parts that match your metadata search query.
	Single Rule Report	Open the Single Rule Report dialog box. Use the Single Rule Report dialog box to generate metadata search conditions and to specify how you want matched and unmatched parts displayed.
	Multiple Rules Report	Open the Multiple Rules Report dialog box. Use the Multiple Rules Report dialog box to generate multiple metadata search conditions.
	Results Legend	Display detailed information about completed visual reports. This information includes details as the current rule name and scope, names of parts, and the attributes and values used in the report.
	Report Explore	Open the Report Explore dialog box. Use the Report Explore dialog box to set options for how you view the results of your visual report.

	Use this option	To do this
	Reset Appearance	Discard visual changes and return to the original visual representation.
	Preferences	Specify the default options for various visual reports preferences.

Overview of visual reports using embedded viewers

Embedded viewers support **Visible parts** for all **Input**. Other input options are unavailable. Embedded viewers do not support cacheless search features, and these viewers only support local scoping.

Note:

Both the Teamcenter and NX applications create and save visual report definition files (.vpx). However, these two visual report definition files are not compatible and definition reports created in one application cannot be opened in the other application. An icon identifies the application that created the report definition file.

From Teamcenter, you can easily work with visualization-based visual reports. You cannot view NX-based visual reports.

Lifecycle Visualization

- Visual reports are saved in a local file system as a **.vpx** file.
- You can save the report results to My Teamcenter as a session.
- You can also save the report results to My Teamcenter as a PLM XML dataset, though this dataset displays only the visual report results, not the visual report data structure.

NX

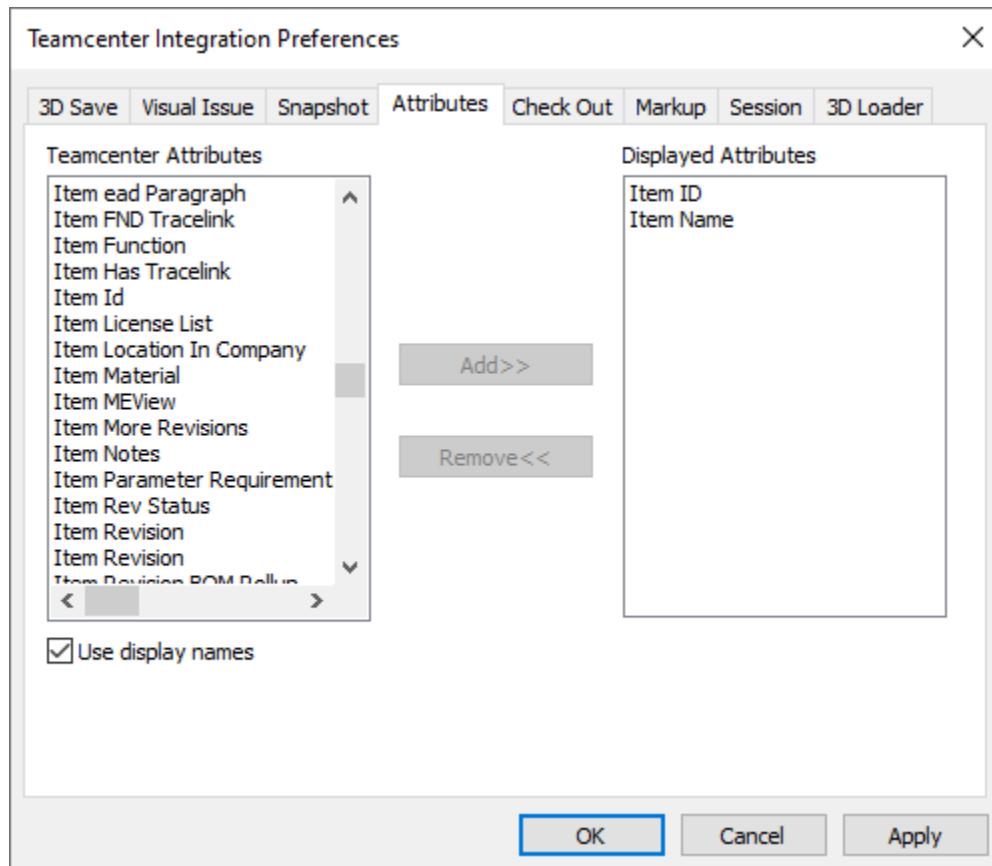
- Visual reports can be saved to the Teamcenter database, but can only be viewed in NX.
- A visual report created in NX and saved to Teamcenter is saved as a report definition (**.vpx**) file. It can only be opened in NX.
- The Teamcenter administrator can also use the Teamcenter Report Builder to create custom, server-side visual reports exclusively for NX applications.

For more information about working with NX visual reports, see the topics regarding the NX Visual Reporting tool delivered with the NX Help Collection.

Limiting BOM line attributes displayed for use in visual reporting

A Teamcenter dataset can include numerous BOM line attributes. The Teamcenter preference **interop_vis_attributes** controls which of these BOM line attributes are displayed for use in visual reporting. The software provides two methods for setting the preference.

- Send your dataset to Lifecycle Viewer and use the **Teamcenter Integration Preferences** dialog box **Attributes** tab to select which attributes to display. Your choices populate the Teamcenter preference **interop_vis_attributes**.



- In Teamcenter, navigate to **Edit→Options** and set the Teamcenter preference **interop_vis_attributes**.

Note:

Teamcenter administrators can control which BOM line attributes are available to all staff by configuring the Teamcenter preference **interop_vis_attributes_admin_filter**.

Visual report options in Teamcenter embedded viewers

For Teamcenter embedded viewers, it may be useful to configure visual report definition files (.vpx) so you can search for them at a later time. You can open report definition files from Teamcenter Classification, My Teamcenter, or the **Newstuff** folder. To determine how visual report definition files are

searched, use the new Teamcenter preference, **VisualReport_Use_Classification**. Set this preference to **True** if you want to use Classification to search for and open visual reports. Set this preference to **False** if you want to use My Teamcenter to search for and open a visual report.

You can also save the visual report to the **Newstuff** folder. From the **Newstuff** folder you can also manually classify the dataset.

Using Global Data Access (GDA)

When you are working in the Teamcenter standalone application viewer and Lifecycle Viewer, if select **Load** from the visual report tool the **Global Data Access (GDA)** is displayed. Use the **GDA** dialog to search for .vpx files by searching the local file system, Teamcenter, or a Teamcenter database.

Creating visual reports

Overview of visual report expressions and rules

Use expressions to set visual report conditions that provide the results you want. Visual reports use standard regular expressions to define Boolean syntax. An expression consists of one or more sets of terms and each term is a property value comparison. For example, if you need a visual report that displays parts that weigh less the 2.5 kg and that were designed by Joe, the expression is **Name=Joe AND Weight<2.5**.

For visual reports, expression rules and definitions include the following:

- Special characters and operators
- Details about condition expressions
- Pattern strings
- Date formatting
- Meta-variables

Special characters and operators

You use regular expressions to specify the pattern used in a search. The following table defines common special characters and operators that you can use in your regular expression syntax.

Use this	To do this
Asterisk (*) special character	Match zero or more characters.
Question mark (?) special character	Match zero or one character.
Backslash (\) special character	Interpret the next character literally. Example: The pattern <code>foo*</code> matches <code>foo*</code>, but does not match <code>food</code>.
Double quotes (") special character	Delimit strings. Special characters inside the quotation marks are treated as special characters. Example: The pattern <code>"John Doe"</code> matches <code>John Doe</code>, <code>John Doe\</code>, and <code>John Doe*</code>.
Single quote (') special character	Delimit strings. Special characters inside the quotation marks are treated as literal characters. Example: The pattern <code>'John*'</code> matches <code>John*</code> but not <code>John</code> or <code>John Doe</code>.
Comparison operator	Compare the left side property to the right side property. Note: For more details, see details about condition expressions.
Logical operator	Combine multiple terms into a complete expression. Note: You can type logical operators as text or symbols. If you type them as text, make sure you use capital letters. Logical operators include <code>AND</code> & <code>OR</code> <code>NOT</code> !. Example: <code>Name=Joe AND Color=red</code> <code>Name=Joe & Color=red</code>

Use this	To do this
	Color=green OR Color=red Color=green Color=red Name=Bill AND NOT (Weight<2.5 OR Color=green)
Parentheses [()] special character	Change the order of precedence or use them to group logical terms together.
Dollar sign (\$) operator	The meta-variable operator.

Details about condition expressions

The basic syntax form for conditional expression interpretation is:

<key> <op> <value>

where <key> (interpreted as the name of the property) and <value> (interpreted as the target value of the property) are character strings and <op> is one of the following comparison operators:

<	Less than
>	Greater than
=	Equal to or matches the value
<=	Less than or equal to
>=	Greater than or equal to
!=	Not equal to or does not match

Three basic value types are supported:

- Date
- Number (integer or floating point)
- String

Note:

- Simple string values are only compared with = (equal to) or != (not equal to) operators.
- If you compare simple string values with less than or greater than operators, the expression evaluates to **false**.

- For date type values, you can use + or – and an integer to add or subtract days, for example [Today]-3.

Pattern strings

A pattern string is one where the expression contains special characters * or ? that are not escaped in the expression. Use pattern strings to identify partial string matches. You can type the special characters on either the left or right side of the operator.

Example:

Color = bl*	Matches the color blue and black, but not navy-blue
* = John*	Matches Owner=John and Author=Johnson, Bill
Name = ?il*	Matches Name=Bill and Name=William, but not Name=Philip

Date formatting

Visual reports support several date formats. These formats are not influenced by the locale in which the application is running. Visual reports support three date field separation characters: slash (/), dash (–), and period (.).

Example:

For most U.S. English locales mm/dd/yyyy

For many Asian-Pacific locales and ISO 8601 yyyy-mm-dd

For many European locales dd.mm.yyyy

If required, you can also include a time specification. The time is separated from the date with either a colon (:), dash (–), or space (). But, the time syntax is always hh:mm:ss.

Meta-variables

When you want to test the value of one property against the value of another property you can use meta-variable syntax. Use the dollar sign (\$) to indicate the meta-variable.

Example:

WEIGHT < \$TARGET_WEIGHT

This means the visual report evaluates each part for the WEIGHT property and the value of the TARGET_WEIGHT property and then compares these values. The report then displays parts that weigh less than the target weight.

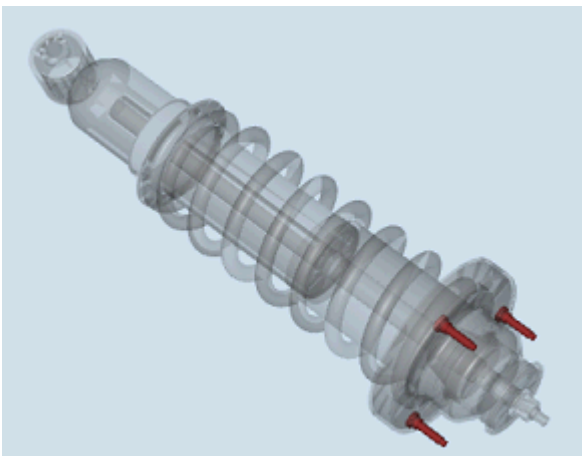
Create a quick visual report

Use **Quick Report** to generate a visual report of components with attribute keys and values that match a specified text expression.

Example:

In this example, a designer wants a visual report of *shock_absorber.jt* parts where the part name starts with st*.

1. On the **3D Visual Report** toolbar, click **Quick Report** .
2. In the **Quick Report** dialog box, type "**part name**" = st*" and click **Apply**.



All parts where the part name starts with st are displayed in the default matched parts material (dark red). All other parts are displayed in the default unmatched material (transparent gray).

Tip:

You can also use the **Find** command to easily create a similar report.

1. On the **3D Visual Report** toolbar, click **Quick Report** .

The **Quick Report** dialog box opens.

2. Type your visual report expression in the text section of the **Quick Report** dialog box.
3. Click **Apply** to generate the visual report and display the results in the Viewing window.
4. (Optional) Click **Preference** to open the **default preferences** options.

In the **Preference** dialog box, you can specify the default colors for matched and unmatched parts. You can also change how the application searches for strings. Select either case insensitive (the default) or case sensitive.

5. (Optional) Click **Save As** to open the **Save Visual Report File** dialog box and save the quick report as a .vpx file.

The quick report name also appears in the **Reports** section of the **Apply Report** dialog box. This dialog box supports **loading and managing** saved visual reports.

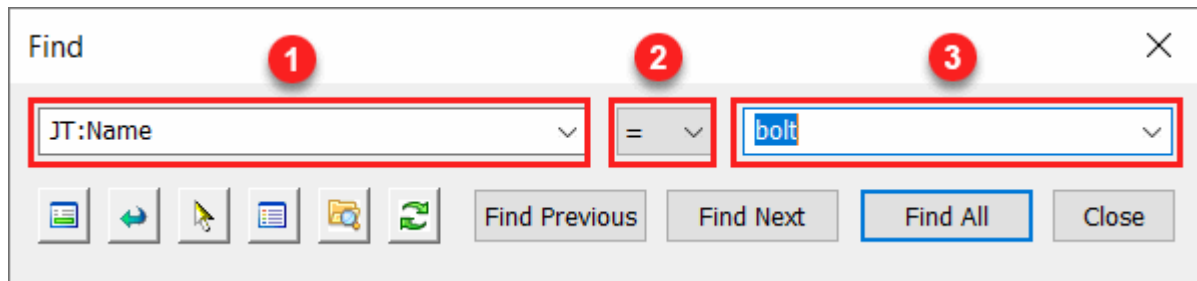
Find parts based on a property value

When Visual Reporting is installed, you can use the advanced **Find** command to find parts and assemblies with property values that match a search pattern and then apply an action to them.

Example:

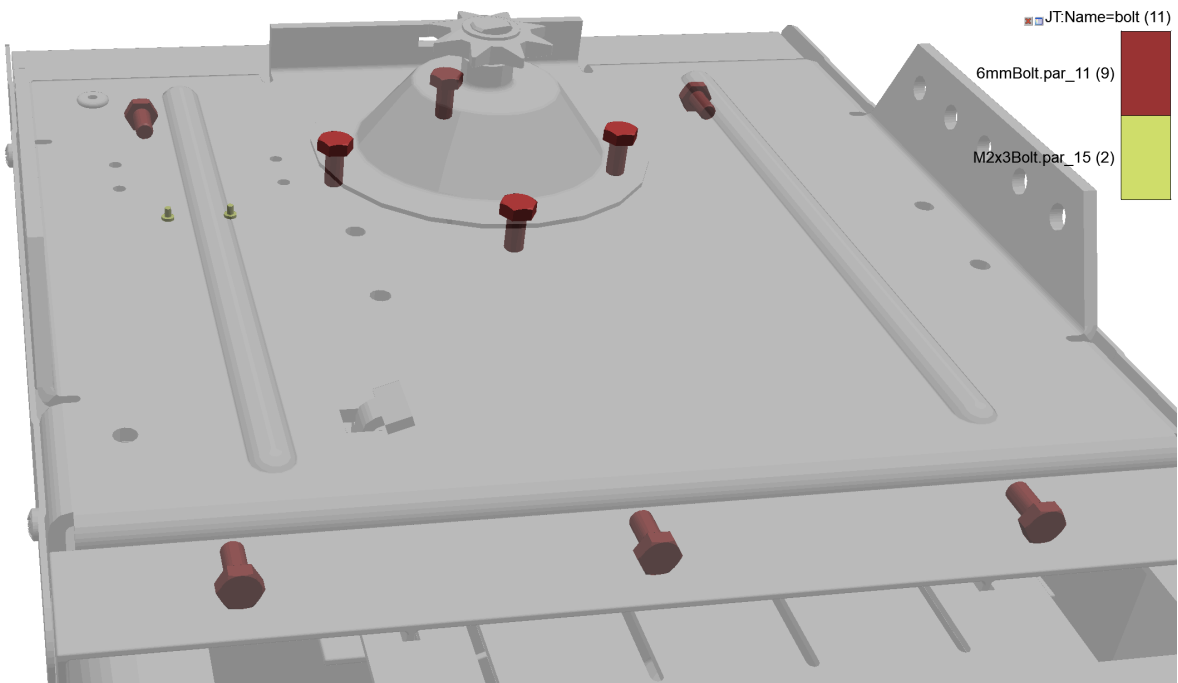
In this example, a designer wants to find all bolts in a model.

1. In an empty area of the Viewing window, right-click and choose **Find**, or choose **Home** tab→**Visual Report** group→**Find** .
2. In the **Find** dialog box, specify **JT:Name = bolt**.



1) Property 2) Condition 3) Value

3. Click **Find All**.



All parts with the JT attribute **Name** where the value includes "bolt" are found and an action can be applied, such as selecting all matching parts. In this case, occurrences of parts **6mmBolt.par_11** and **M2x3Bolt.par_15** match the search pattern, and each occurrence group is colored with a group color. Parts that do not match the pattern are colored transparent gray.

Tip:

Create a snapshot or a group to save results of advanced **Find** actions.

Procedure

- 1. Start the command in one of these ways:

For this initial property availability	Start the command this way
Properties and values from the first 100 nodes in the product structure	In the Viewing window, right-click an empty area and choose Find , or choose Home tab→ Visual Report group→ Find  .
Properties and values from a particular node or part, with the ability to peruse and copy properties and values from the Properties dialog box for the node or part	Right-click a node or part and choose Properties , and then in the Properties dialog box, click Find . While both dialog boxes are open, you can click an attribute name in the Properties dialog box and the property and its value are copied to the Find dialog box.

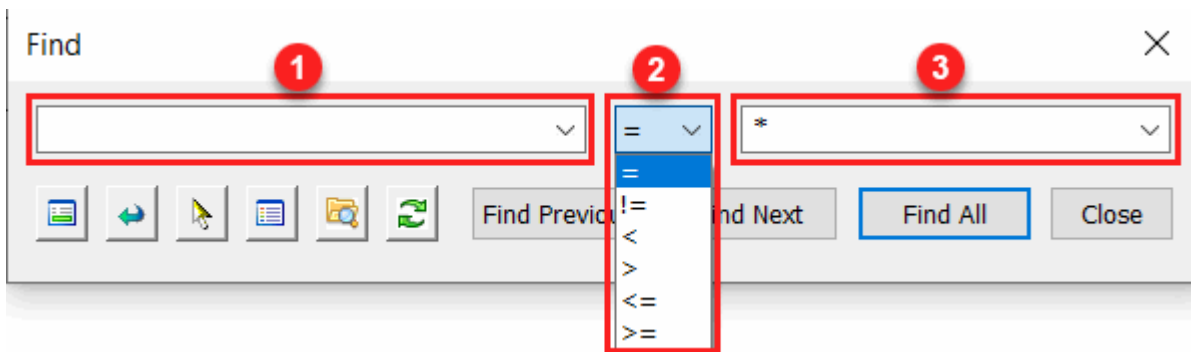
For this initial property availability	Start the command this way
	Mouse navigation modes are not available while the Properties dialog box is open. When the Find dialog box is opened from the Properties dialog box, closing the Properties dialog box also ends the Find command.

2. As needed, specify what you want to do with the Find operation. Click **Preferences**  to open the **Find Preferences** options dialog box.

To do this	Select this Option
Limit parts to include in the search.	Input Choose an option from the list.
Choose the action to apply to the search results.	Action • Show only (Loads matching parts) • Show additive (Loads matching parts) • Hide • Select only • Select additive • Deselect • Change appearance (Loads matching parts) You can click More Options to specify how the appearance of matched and unmatched results should change.
Include assembly nodes in the results, in addition to parts.	Include assembly nodes
Zoom the view window to the results of the search.	Zoom to matching results

To do this	Select this Option
Match only property values that have only the specified pattern characters.	Match whole word only
Match only property values where the character case matches the specified pattern character case.	Match case
Create a visual report of the results.	Click More Options to open the Visual Report Preferences dialog box and specify options for creating a visual report of the results.

3. In the **Find** dialog box, specify parameters for the property pattern you want to find.



For this pattern parameter	Do this
1) Property	Enter or select the property for the search pattern. Note: To refresh the list from properties of the currently selected nodes or parts, click Refresh Attributes List . If no nodes or parts were selected, a message appears prompting you to select a node or part.
2) Condition	Select the condition for comparing the property value.

For this pattern parameter	Do this
	Note: For text values, only the = (contains) and != (doesn't contain) conditions are valid.
3) Value	Enter or select the value to search. The value can be either numbers or text. Note: When the value box (3) contains an asterisk (*), all parts and assemblies that have the selected property match the search pattern.

4. Click a **Find** command to find the matching occurrences and apply the specified actions.

- **Find Previous** — returns focus to a previous matching occurrence
- **Find Next** — sets focus to the next matching occurrence
- **Find All** — applies actions to all matching occurrences

Additional options

To do this	Click this command
Clear the applied action(s).	Clear  If no visual report was applied to the assembly, a message appears prompting you to reset the appearance to the default materials. If a visual report was applied to the assembly, the appearance is reset to the state before the report was applied.
Select all matching parts found by the search.	Select Matching Parts 
Display an interactive legend for the search results.	Results Legend 

To do this	Click this command
	For information about manipulating display of results in the Results Legend dialog box and in the Viewing window, see Examine results of the visual report .
Cycle through reports for properties or values (available in Windows OS only).	Explore Mode  and then: - Click either the property box or the value box to set focus on that parameter. - Press the up or down arrow key to choose the next item in the parameter list. Note: If focus is on the property box, when you press an arrow key to choose a new property, the value box automatically is set to asterisk (*) so that all values for the property are matched.
Refresh the properties and values lists.	Refresh Attributes List  Refreshes the lists from properties of the currently selected nodes or parts. If no nodes or parts were selected, a message appears prompting you to select a node or part.

Create a single rule visual report using local scoping

When your data is saved in the local file system you use local (client-side) scoping to generate visual report rules. The single rule report is used to search the parts of assemblies by composing a query for specific attributes and values.

- Open a 3D assembly model.
- On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
- (Optional) To create a new visual report using a previously saved report as a template, click **Load** and open the saved report.
- In the **Single Rule Report** dialog box, in the **Report Scope** section, click **Use local** scoping.

Single Rule Report

New Report

Report

Input: Visible parts

Action: Change appearance

Report Scope

☒ Use local scoping ☐ Use Teamcenter scoping

Source:Attribute... = Value...

☐ Case sensitive

Matched Results Appearance

☒ Single appearance ☐ Color ramp

Transparency: 0.00 0 1

Display mode:

Select material from palette:

Unmatched Results Appearance

☐ Keep original appearance

Transparency: 0.75 0 1

Display mode:

Select material from palette:

5. In the **Report** section, specify the **Input** (parts to analyze) and **Action** (what to do to).
 - a. Do one of the following:

To analyze	Do this
All visible parts	From the Input list, choose Visible parts .
All selected parts	From the Input list, choose Selected parts .

To analyze	Do this
All parts	From the Input list, choose All parts .
Matched occurrences from a previously applied report	Do the following: - Click Previous. - In the Previous Report dialog box, check the name of the report, the rule(s), and rule scope. (Optional) In the Matched Results section, click Select Parts to highlight parts that match the scope of the rule. For easy navigation, matched parts are also highlighted in the Assembly view. (Optional) In the Matched Results section, click Show Parts to only display the parts that match the scope of this rule. (Optional) Use the parts identified by either Select Parts or Show Parts to generate a second report that searches additional details and creates a visual report consistent with the new rules. Example: You need to provide two visual reports. One report will search for all parts release on a certain date (or relative date like [Today]-3); the second report displays designers who are overdue with their designs. You set the report scope for the first condition, all parts released on a certain date, and generate the visual report. Then set the report scope for the second condition, designers who are overdue, and then click Previous. Choose to apply the report scope of the second report to either Select Parts or Show Parts.

b. From the **Action** list, select one of the following:

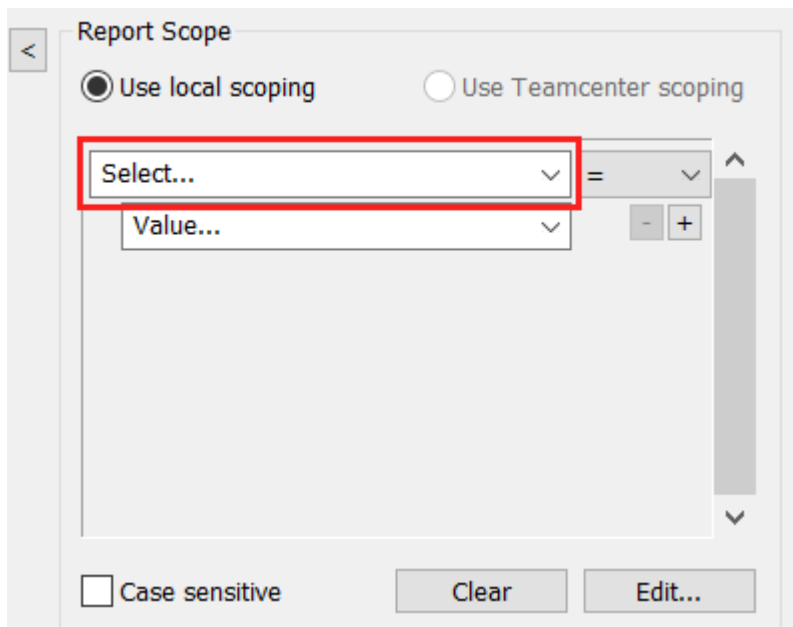
- **Change appearance**
- **Show only**
- **Show additive**
- **Hide**

- **Select only**
- **Select additive**
- **Deselect**

6. In the **Report Scope** section, specify the attribute, comparison operator, and value.

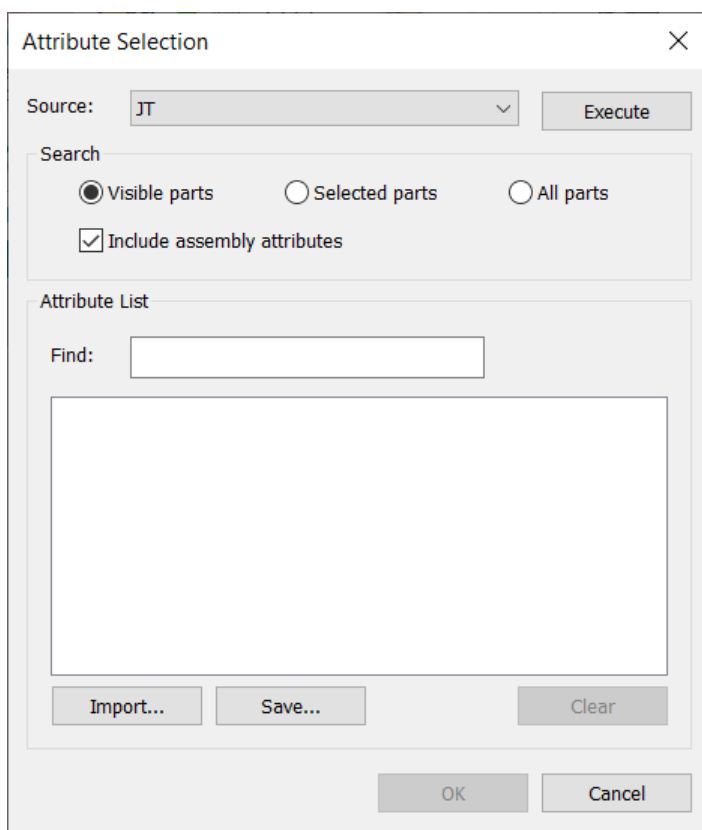
For example: Material = Steel or SignOff_Date > 2015/01/01

a. In the **Source:Attribute** box, type the attribute name or choose **Select**.



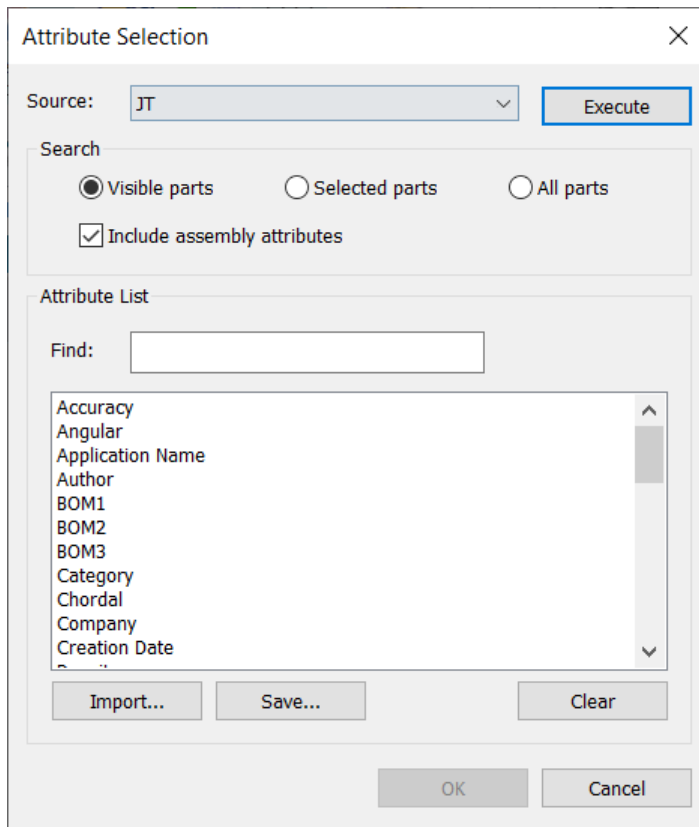
If you choose **Select**, the **Attribute Selection** dialog box appears.

b. Make sure **Source** is set to **JT**.

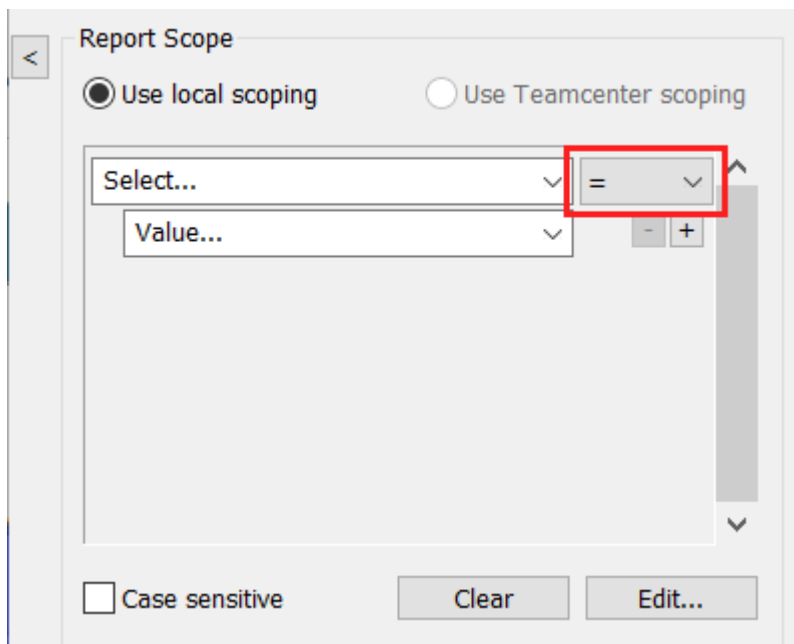


The image shows a software dialog box titled "Attribute Selection" with a close button (X) in the top right corner. The dialog is organized into several sections. At the top, there is a "Source:" label followed by a dropdown menu showing "JT" and an "Execute" button. Below this is a "Search" section containing three radio buttons: "Visible parts" (which is selected), "Selected parts", and "All parts". There is also a checked checkbox labeled "Include assembly attributes". The main area of the dialog is the "Attribute List" section, which includes a "Find:" text input field and a large, empty rectangular list box. At the bottom of the "Attribute List" section are three buttons: "Import...", "Save...", and "Clear". At the very bottom of the dialog are "OK" and "Cancel" buttons.

- c. Click **Execute** to generate a list of attributes for the assembly model.



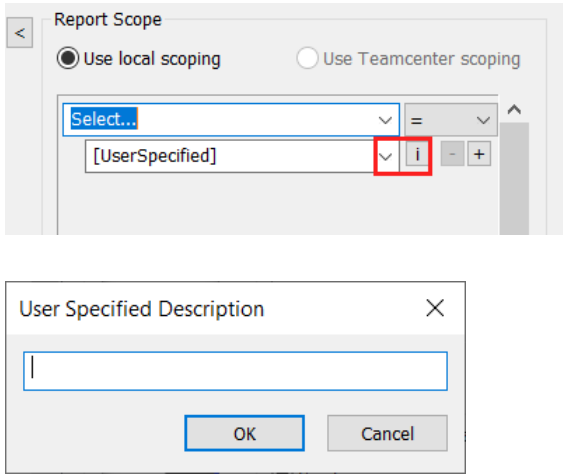
- d. Highlight the attribute and click **OK**.
- e. From the comparison operator list, select the operator.

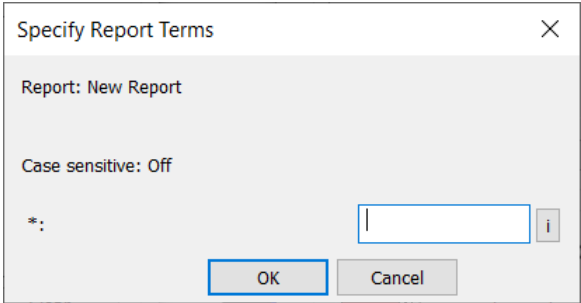


Some operators apply only to numeric or date attributes.

- f. Do one of the following to specify the value:

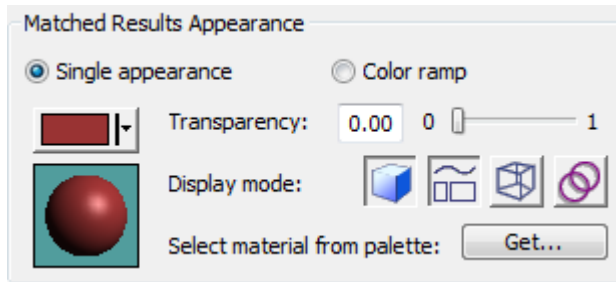
To	Do this										
Manually specify the value	Type the value in the Value box.										
Select the value from a generated list of values	A. Do the following: <table> <tr> <th>To display values for</th><th>Do this</th></tr> <tr> <td>One or more parts</td><td> i. Highlight the parts. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate. </td></tr> <tr> <td>One or more subassemblies</td><td> i. Highlight the subassemblies. ii. Choose Select. iii. In the Attribute Values dialog box, click Regenerate. </td></tr> <tr> <td>One or more assemblies and all of their subassemblies and parts</td><td> i. Highlight the assemblies. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate. </td></tr> <tr> <td>All parts and subassemblies in a model</td><td> i. Highlight the model node. ii. Choose Select. iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate. </td></tr> </table>	To display values for	Do this	One or more parts	i. Highlight the parts. ii. Choose Select . iii. In the Attribute Values dialog box, click Regenerate .	One or more subassemblies	i. Highlight the subassemblies. ii. Choose Select . iii. In the Attribute Values dialog box, click Regenerate .	One or more assemblies and all of their subassemblies and parts	i. Highlight the assemblies. ii. Choose Select . iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate .	All parts and subassemblies in a model	i. Highlight the model node. ii. Choose Select . iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate .
To display values for	Do this										
One or more parts	i. Highlight the parts. ii. Choose Select . iii. In the Attribute Values dialog box, click Regenerate .										
One or more subassemblies	i. Highlight the subassemblies. ii. Choose Select . iii. In the Attribute Values dialog box, click Regenerate .										
One or more assemblies and all of their subassemblies and parts	i. Highlight the assemblies. ii. Choose Select . iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate .										
All parts and subassemblies in a model	i. Highlight the model node. ii. Choose Select . iii. In the Attribute Values dialog box, select the Show values for all children check box. iv. Click Regenerate .										

To	Do this
	B. Select the attribute value and click OK.
Specify a relative date	From the Value list, select one of the following: • [Today] • [Yesterday] • [PastWeek] • [PastMonth] • [Tomorrow] • [NextWeek] • [NextMonth] (Optional) Type + or – and an integer to add or subtract days. For example, specify [Today]-3.
Have the system prompt you to enter a value when you apply the report	Do the following: A. From the Value list, choose [UserSpecified]. B. (Optional) To enter a description of the attribute, click [i] and type the description in the User Specified Description dialog box.  When you apply the visual report, the Specify Report Terms dialog box appears and you can type the attribute value in the text box.

To	Do this
	 You can click [i] to display the attribute description.

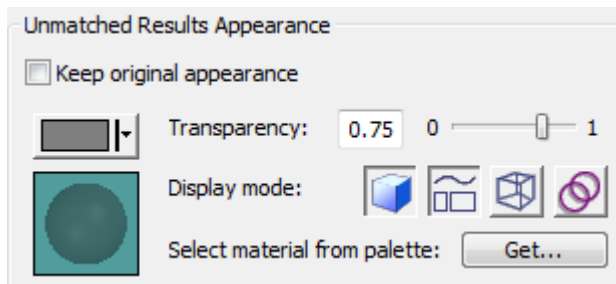
- g. (Optional) Select the **Case sensitive** check box to create string searches that are case sensitive.
- h. (Optional) Click **Edit** to **manually enter** the attribute, operator, and value string.
7. (Optional) Click + in the **Report Scope** section to specify an AND or OR attribute, operator, and value string.
8. (If the report **Action** is **Change Appearance**) In the **Matched Results Appearance** section, specify how you want to display parts that match your visual report scope.
 - a. Select a color pattern by doing one of the following:
 - To specify a single color for matched parts, select **Single appearance** and choose a color from the list.
 - To specify a color range, select **Color ramp**.
 - b. (Optional) If you specify a single color, in **Transparency**, move the slider to specify how much color transparency you want.
 - c. From **Display mode**, click one or more display styles.
 - d. (Optional) Click **Get** to select a material from the Materials palette.

This allows you to use advanced materials with **textures**.



9. (If the report **Action** is **Change Appearance**) In the **Unmatched Results Appearance** section, specify how you want to display parts that do not match your visual report query.
 - a. Select **Keep original appearance** to display unmatched parts in their original color.
 - b. Select a color from the list and set the **Transparency**.
 - c. From **Display mode**, click one or more display styles.
 - d. (Optional) Click **Get** to select a material from the Materials palette.

This allows you to use advanced materials with **textures**.

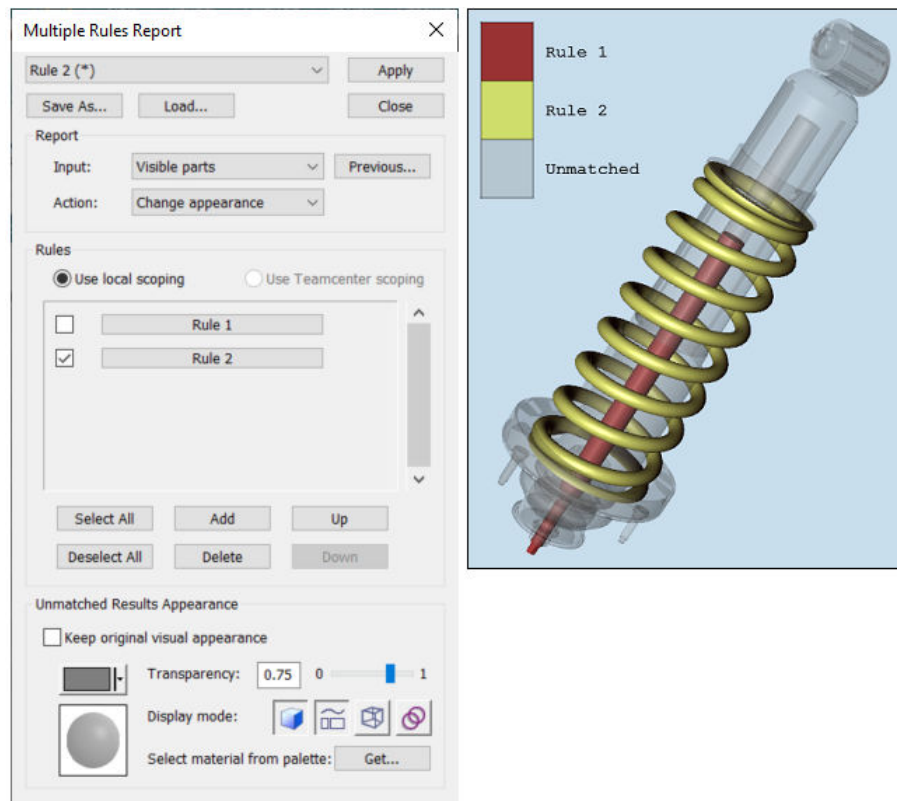


10. Click **Save As** or **Apply**.

Create a visual report containing multiple rules using local scoping

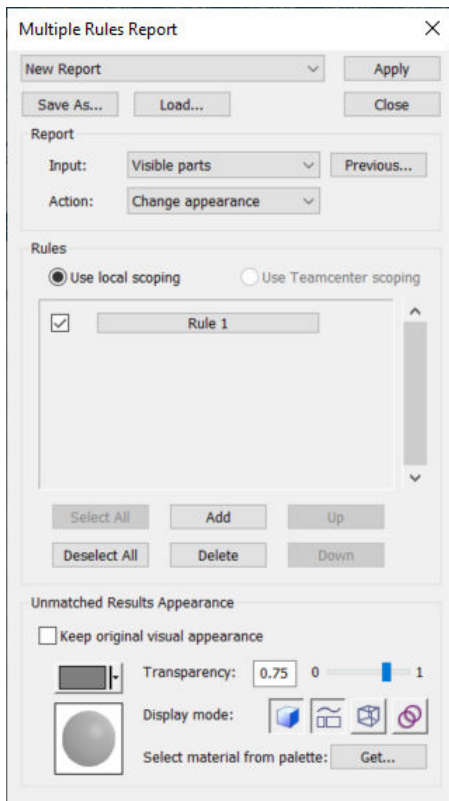
Use multiple rules reports when you need to query your assemblies with several rule constraints. When you use multiple rules to generate visual reports, the process is nearly the same as it is for generating single rule reports. You add several single rules using the **Single Rule Editor** dialog box that includes the **Report Scope** and **Matched Results Appearance** sections of the **Single Rule Report**.

Example:



1. On the **Visual Report** toolbar, click **Visual Report** , and then click **Multiple Rules Report** .

The **Multiple Rules Report** dialog box appears.



2. (Optional) To create a new visual report using a previously saved report as a template, click **Load** and open the saved report.
3. In the **Report** section, do the following:
 - a. Do one of the following:

To analyze	Do this
All visible parts	From the Input list, choose Visible parts .
All selected parts	From the Input list, choose Selected parts .
All parts	From the Input list, choose All parts .
Matched occurrences from a previously applied report	Do the following: A. Click Previous. B. In the Previous Report dialog box, check the name of the report, the rule(s), and rule scope. C. (Optional) In the Matched Results section, click Select Parts to highlight parts that match the scope of the rule.

To analyze	Do this
	For easy navigation, matched parts are also highlighted in the Assembly view. D. (Optional) In the Matched Results section, click Show Parts to only display the parts that match the scope of this rule. E. (Optional) Use the parts identified by either Select Parts or Show Parts to generate a second report that searches additional details and creates a visual report consistent with the new rules. Example: You need to provide two visual reports. One report will search for all parts release on a certain date (or relative date like [Today]-3); the second report displays designers who are overdue with their designs. You set the report scope for the first condition, all parts released on a certain date, and generate the visual report. Then set the report scope for the second condition, designers who are overdue, and then click Previous. Choose to apply the report scope of the second report to either Select Parts or Show Parts.

- b. In **Action**, select how you want to display the matched occurrences.

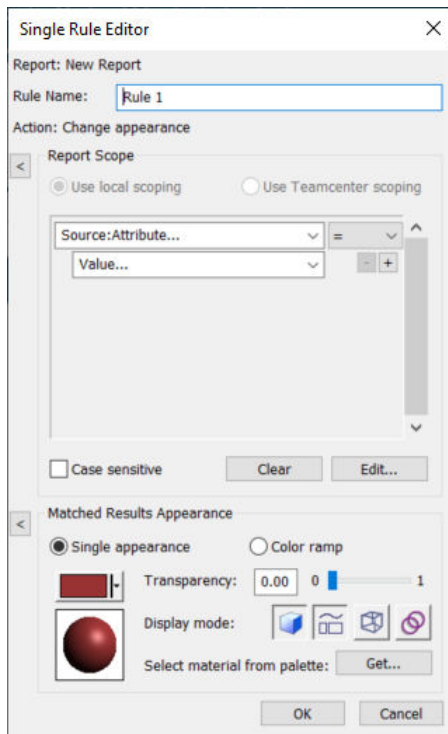
Note:

Only the **Change appearance** option supports editing **Matched Results Appearance** and **Unmatched Results Appearance**.

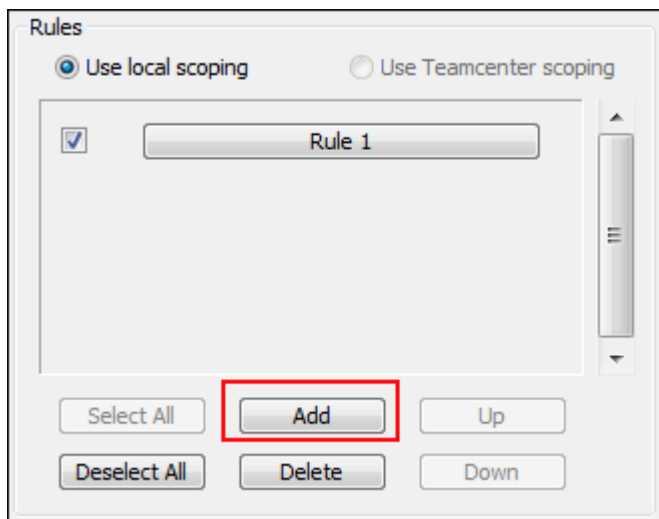
4. In the **Rules** section, create or modify the multiple rules for this report.

- a. Click **Rule 1**.

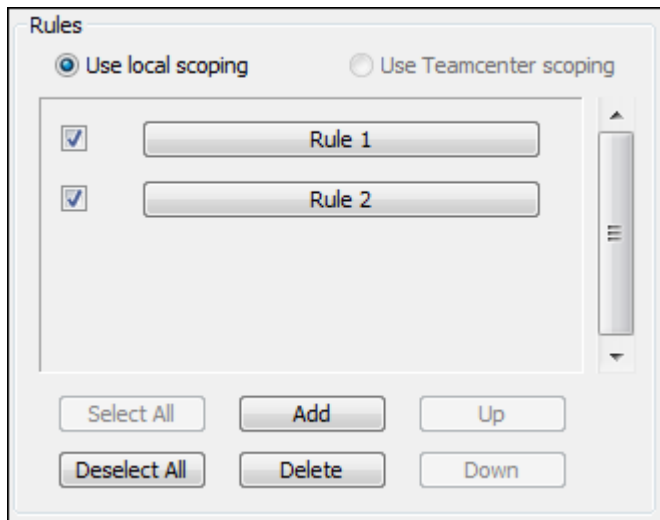
The **Single Rule Editor** dialog box appears.



- b. Configure the scope for the first rule exactly as you configure a **single rule report**, and then click **OK**.
- c. Click **Add** to add a second rule.



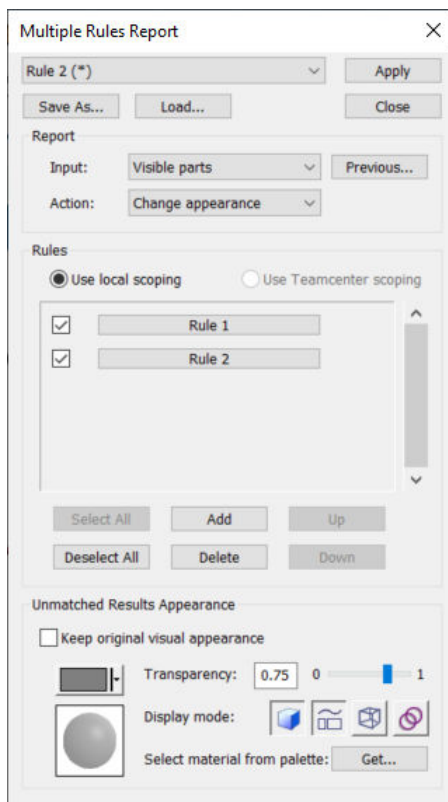
Rule 2 appears.



- d. Click **Rule 2**.

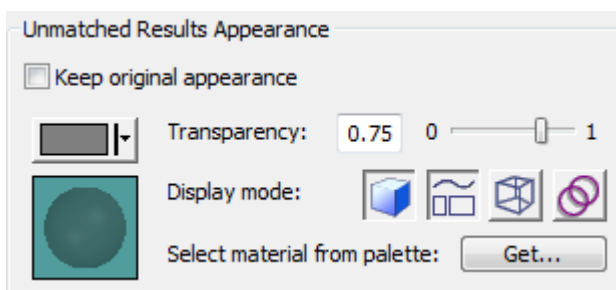
The **Single Rule Editor** appears again.

- e. Configure the scope for the second rule and click **OK**.



5. (Optional) In the **Rules** section, you can modify the rules (select, delete, and move) clicking any of the following:

- **Select All**
 - **Deselect All**
 - **Delete**
 - **Up**
 - **Down**
6. In the **Unmatched Results Appearance** section, specify how you want to display parts that do not match your visual report query.



- a. Select **Keep original appearance** to display unmatched parts in their initial color.
- b. Clear **Keep original appearance** and select a single color from the list.
- c. (Optional) In **Transparency**, move the slider to specify how much color transparency you want displayed.
- d. From **Display mode**, click one or more display styles.

You can test each mode or combination of modes. Click one or more display modes and then click **Apply**. The results are immediately displayed in the Viewing window.

- e. (Optional) Click **Get** to open the **Materials**, and then select a material to quickly display parts in a color of your choice.

Note:

For large assemblies, a visual report may display numerous colors for matched and unmatched parts. These colors may blend so closely that the results could be misinterpreted. In this situation, it may be useful to specify a known unmatched material color before you apply the report.

7. Click **Save As** or **Apply**.

Create visual reports using Teamcenter scoping

You can use Teamcenter scoping for visual reports when the assembly is loaded into the viewer from Teamcenter. Teamcenter scoping utilizes the Teamcenter cacheless search capabilities to provide access to attributes and values directly from the Teamcenter server.

Note:

The following describes using Teamcenter scoping for Single Rule Reports, but the process is the same process for Multiple Rule Reports.

1. Open the assembly in Teamcenter and select either **Start\Open in Lifecycle Visualization**, or right-click the assembly and choose **Send To→Lifecycle Viewer**.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then **Single Rule Report** .

In the **Single Rule Support** dialog box, **Use Teamcenter scoping** is automatically selected when the data is from Teamcenter.

Single Rule Report

New Report ▾ Apply

Save As... Load... Close

Report

Input: All parts ▾ Previous...

Action: Change appearance ▾

Report Scope

☐ Use local scoping ☒ Use Teamcenter scoping

Source: Attribute... ▾ = ▾

Value... ▾ - +

☐ Case sensitive Clear Edit...

Matched Results Appearance

☒ Single appearance ☐ Color ramp

Transparency: 0.00 0 1

Display mode:

Select material from palette: Get...

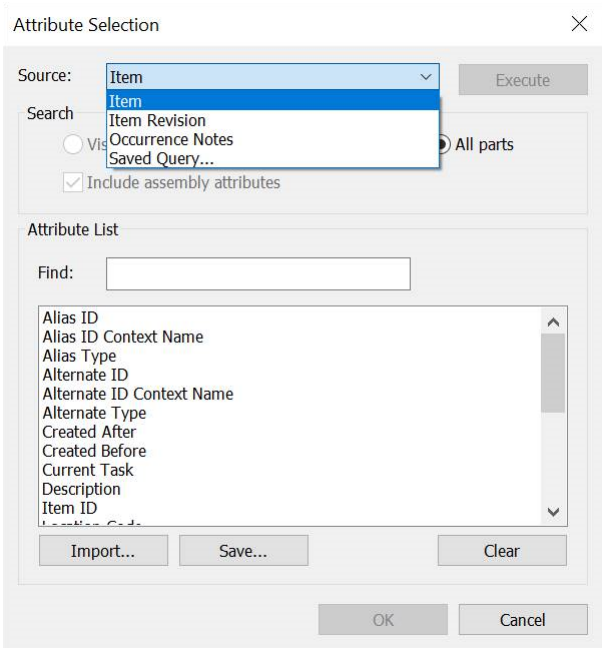
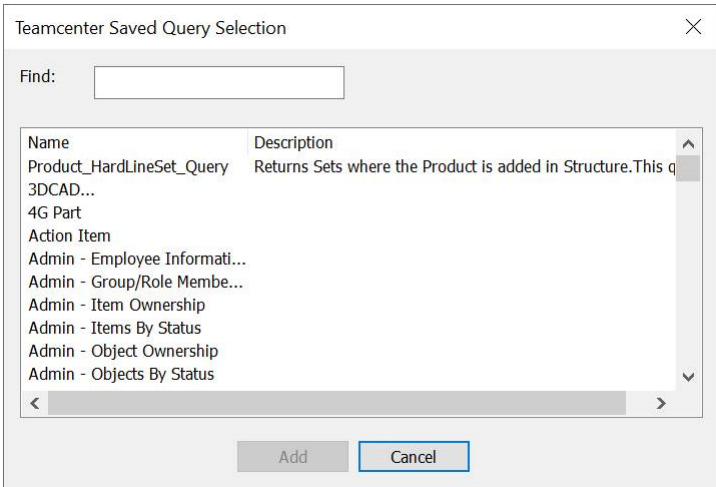
Unmatched Results Appearance

Note:

Teamcenter scoping is not supported with Workset or 4GD workset data.

3. Select **Source:Attribute**.
4. In the **Attribute Selection** dialog box, do one of the following:

To	Do this
Create a new query	a. From the Source list, select Item , Item Revision , or Occurrence Notes .

To	Do this
	 b. Select an attribute and click OK.
Use a save query	a. From the Source list, select Saved Query b. In the Teamcenter Saved Query Selection dialog box, select from the list queries that have been saved to the server. 

5. For details about configuring a **Single Rule Report**, see [Create a single rule visual report using local scoping](#).

Note:

For Teamcenter scoping, only AND is supported.

6. For details about configuring a **Multiple Rule Report**, see [Create a visual report containing multiple rules using local scoping](#)
7. When you are finished, click **Apply**.

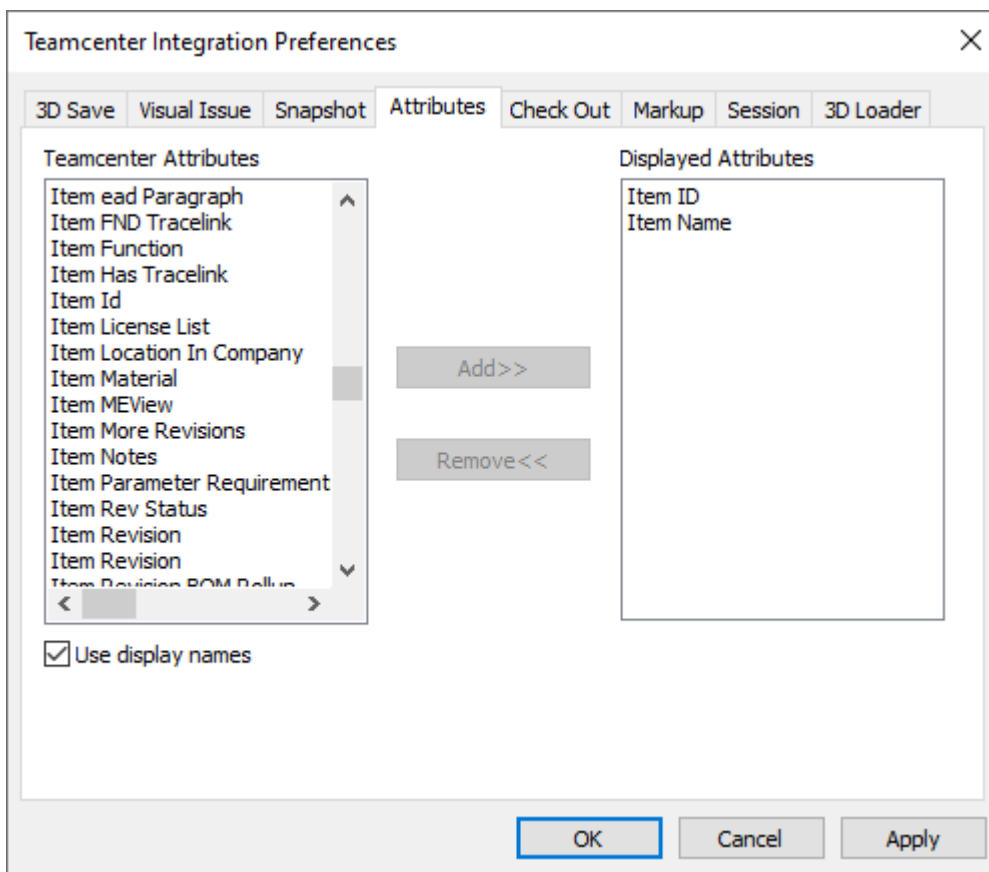
The results of your visual report are displayed in the Viewing window.

Create visual reports using Teamcenter structure lines

You can create client-side visual reports using Teamcenter structure cached on your local machine. This enables you to create visual reports using data not supported with Teamcenter scoping (cacheless searches), such as Worksets or 4GD worksets.

Prior to creating a visual report, you may need to specify the BOM line attributes to be loaded. If you are interested in 4G Designer element attributes, you do not need to specify the attributes to be loaded.

1. Send an object associated with 3D visualization data into the standalone viewer or the Lifecycle Viewer.
2. In the viewer, choose **File** tab→**Preferences**→**Teamcenter Integration**.
3. In the **Teamcenter Integration Preferences** dialog box that appears, select the **Attributes** tab.



4. In the **Teamcenter Attributes** section, highlight the attributes you want to include in the visual report, and then click **Add**.

Selected attribute objects appear in the **Displayed Attribute** section.

5. Click **OK**.
6. Close and reopen the file for the preference change to take effect.
7. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then **Single Rule Report**  or **Multiple Rules Report** .
8. In the **Report Scope** section, make sure **Use local scoping** is selected.
9. Select **Source:Attribute**, and in the **Attribute Selection** dialog box that appears, in **Source** select **Teamcenter Structure Lines**.

Notice the objects selected in **Teamcenter Integration Preferences** also appear in the **Attribute List** section of the **Attribute Selection** dialog box.

10. Click **Execute**.

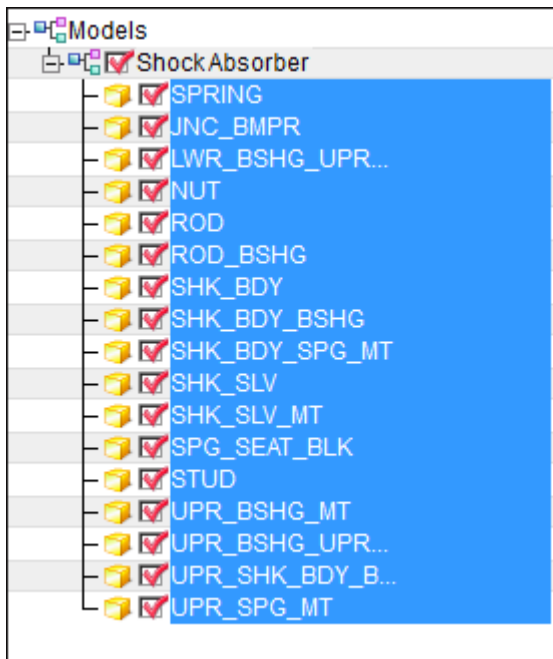
11. For details about configuring a **Single Rule Report**, see [Create a single rule visual report using local scoping](#).
12. For details about configuring a **Multiple Rule Report**, see [Create a visual report containing multiple rules using local scoping](#).

Create visual reports using relative dates

1. Open an assembly.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the Assembly tree, select at least one part.

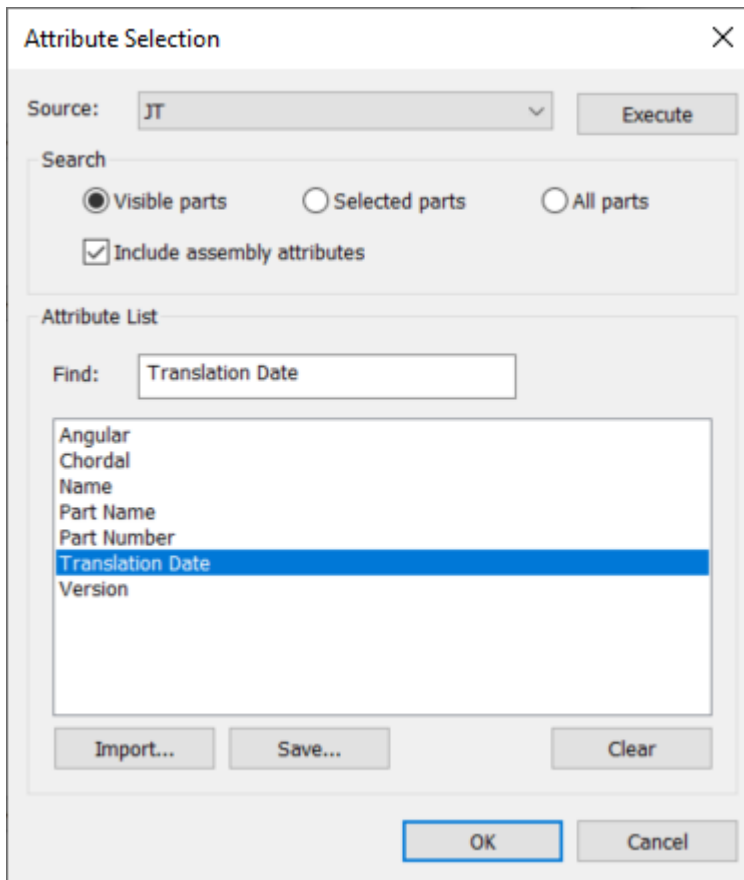
Note:

- If you select all of the parts, you display the entire list of values for this attribute.
- You must select at least one part on the Assembly view. If you do not highlight at least one part, an error message appears.
- If you select all the parts associated with the assembly, all available attribute values appear.



4. In the **Single Rule Report** dialog box, in the **Report Scope** section, choose **Source:Attribute** or **Select**.

5. In the **Attribute Selection** dialog box, to populate the **Attributes List** with the attributes of the selected parts, click **Selected Parts** and **Execute**.
6. Select a date attribute and click **OK**.



7. In the **Report Scope** section of the **Single Rule Report** dialog box, from the **Select** list, choose a condition from the following options:
 - [Today]
 - [Yesterday]
 - [PastWeek]
 - [PastMonth]
 - [Tomorrow]
 - [NextWeek]
 - [NextMonth]

8. (Optional) Type + or – and an integer to add or subtract days. For example, type [Today]-3.
9. In the **Single Rule Report** dialog box, click **Apply**.

If the condition evaluates as specified, the results are displayed in the Viewing window.

10. (Optional) Save the report to compare an instance of this condition in the future.

Comparing visual reports using snapshots

Compare visual reports using snapshots

You can compare the report scope of visual reports over time. For example, you can compare relative dates, version status, and completion status from one point in time to another point in time.

1. Open an assembly.
2. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the **Single Rule Report** dialog box, in the **Report Scope** section, click **Source:Attribute** and then choose **Select**.
4. In the **Attribute Selection** dialog box that appears, click **Execute**, and select an attribute from the list.
5. Click **OK**.
6. Highlight at least one part in the Assembly tree.
7. In the **Single Rule Report** dialog box, in the **Report Scope** section, choose a value from the **Select** list.
8. Click **Apply**.
9. Save the visual report.
10. From the viewer, open the **Snapshots** view and select **Add**.
11. (Optional) In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Reset** .

This step resets the assembly to its original state.

12. Choose **File→Save Session As**, and save the session file.

13. At a later time, open the previously saved session file and load the saved visual report.
14. Follow previous steps to create another snapshot and save the session file again.
15. In the Lifecycle Viewer, on the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
16. In the **Single Rule Report** dialog box, double-click one of the snapshots. Then, double-click the other snapshot.

The visual report results are displayed in the Viewing window. The corresponding visual **Report Scope** appears in the **Single Rule Report** dialog box.

Tip:

See an **example** about comparing visual reports using snapshots.

Example – comparing visual reports

This example creates snapshots of parts with different *version* numbers. The sample file is *shock_absorber.jt*.

You can use this example as a foundation for creating meaningful snapshots and visual reports. For example, you may want to compare when part design is completed, what the status is for each part during each design phase, and so forth.

1. Open Teamcenter lifecycle visualization and load *shock_absorber.jt*. This data is in the *Examples* folder, which is located in the Teamcenter lifecycle visualization installation directory.
2. On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. In the **Single Rule Report** dialog box, set **Report Scope** as (Version = 6).
4. In **Matched Results Appearance**, set the **Single appearance** to green.
5. Click **Apply**.
6. Navigate to the **Snapshots** view and click **Add**.

Note:

You may want to rename the snapshot, for example, *V6 green*.

7. Save this visual report, for example, *test_green.vpx*.

8. Save this session file as *shock_absorber_version6.vf*.
9. Follow the steps above to create and save a second visual report, snapshot, and session file.
10. For the second visual report, in **Report Scope**, type (Version = 2).
11. In **Matched Results Appearance**, set the **Single appearance** to red.
12. Click **Apply**.
13. Navigate to the **Snapshots** view and click **Add**.

Note:

You may want to rename the snapshot, for example, *V2 red*.

14. Save this visual report, for example, *test_red.vpx*.
15. Save this session file as *shock_absorber_version2.vf*.
16. Navigate to the **Snapshots** view and double-click one of the snapshots. Now double-click the other snapshot.

Both the Viewing window and the **Single Rule Report** dialog box display the selected visual report conditions. You can cycle from one snapshot to the other and review the different conditions.

You can cycle from one snapshot to another and review the scope of the visual report. If you created a visual report snapshot one month ago and then create one today, you can compare changes to the model.

Load and manage existing visual reports

You can edit a report, rename a report, and delete a report from the list of saved visual reports.

1. From the viewer, open the model associated with the saved visual report.
2. On the **Visual Report** toolbar, click **Apply Report** .

The **Apply Report** dialog box appears.

3. (Optional) If you have created and saved visual reports and you want to load one or more of them, do the following:
 - a. In the **Management** section click **Load**.

- b. In the **Load Visual Report File** dialog box, highlight the reports that you want to load, and then click **Open**.
4. In the **Reports** section of the **Apply Report** dialog box, highlight the visual report you want to apply and then click **Apply**.

The visual report is applied to the model and it appears in the Viewing window.

5. (Optional) To modify the rules of any saved visual reports, select **Edit**.

Note:

- Depending on the type of report, the **Single Rule Report** or the **Multiple Rules Report** dialog box appears.
- You can save the edited visual report by selecting **Save** in the **Single Rule Report** or the **Multiple Rules Report** dialog box.
- An edited, but unsaved rules report appears with an * in the list.

6. (Optional) To rename the visual report, in the **Single Rule Report** dialog box, click **Save As** and type a new name for this visual report.

Examining visual report results

Explore visual reports by stepping through parts

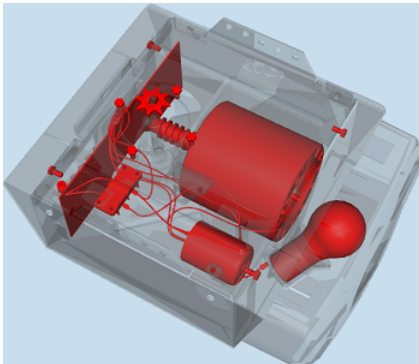
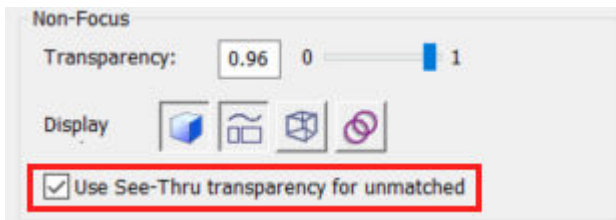
1. From the viewer, open the assembly and apply a visual report of interest.
2. On the **Visual Report** toolbar, click **Report Explore** .
3. In the **Report Explore** dialog box, do the following:
 - a. Make sure the **Report** section displays the visual report you selected.
 - b. In the **Assembly/Part Focus** section, step through matched occurrences using the options **Top** and **Bottom** or, **First**, **Previous**, **Next**, and **Last**.
 - c. (Optional for assemblies only) Select one of the following from the list in **For Assemblies**:
 - **Use assembly attribute** to set the color of the focused assembly based on the matching color of the assembly.
 - **Show matched children** to set the color of the focused assembly based on the color of its children.

- **Use first matched child** to set the color of the focused assembly based on the color of the highest matching child.
- (Optional) In the **Rule Focus** section, clear or select the rule or rules that you want to review.
 - (Optional) in the **Non-Focus** section, set the **Transparency** and **Display mode** for parts that are not in the rule focus.

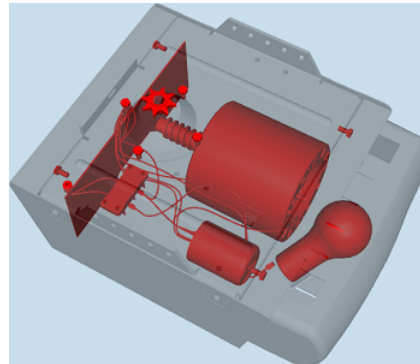
Note:

Use these options to change the display characteristics of unmatched parts because these parts may be obscuring the matched parts that you want to review.

- (Optional) Select the **Use See-Thru transparency for unmatched** check box to view the visual report looking through unmatched parts.



Standard transparency



See-Thru transparency

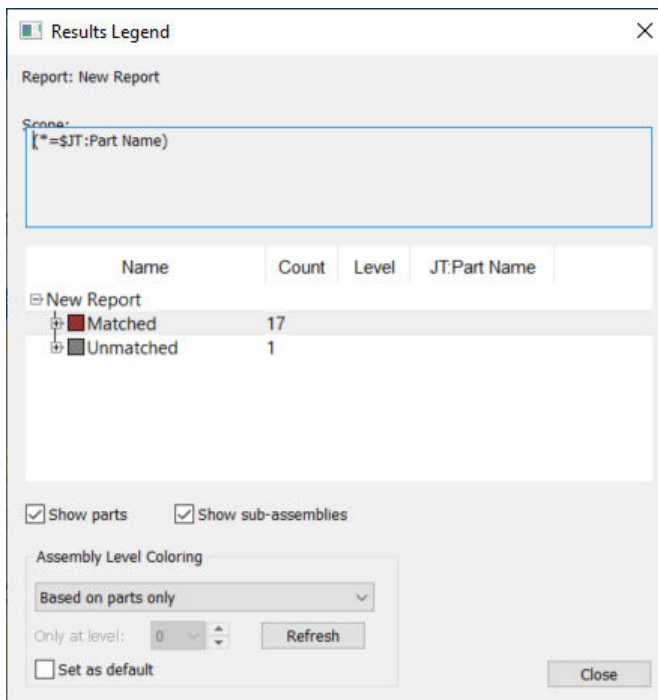
Examine results of the visual report

1. Open an assembly.
2. On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
3. Create the **Report Scope** for this visual report.
4. In the **Single Rule Report** dialog box, click **Apply**.

In the assembly structure, each assembly node and part that matches the search pattern is assigned a unique result color. Appearance of parts in the viewing window is changed based on the result colors and results preferences. You can use options in the **Results Legend** dialog box to manipulate display of results.

5. On the **3D Visual Report** toolbar, select **Results Legend** .

Or, click the **Results Legend** button on the visual report.



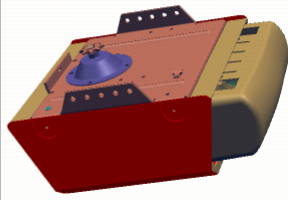
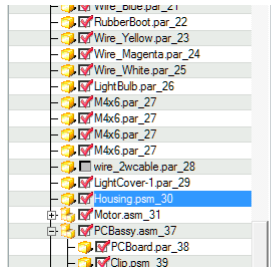
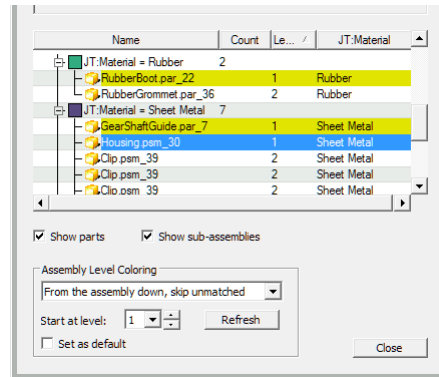
6. Manipulate display of results in the dialog box and in the viewing window.

To	Do this
Show or hide parts and subassemblies in the Results Legend dialog box	Select or clear the check boxes for Show parts and Show subassemblies .
Choose how you want to display result coloring in the viewing window	From the Assembly Level Coloring section, choose an option from the list. Note: If you change any assembly level color setting, be sure to refresh the view by clicking Refresh.

To	Do this	
	Use this option	To do this
	Based on parts only	Display part color based on the part's result color.
	From the parts up, skip unmatched	If a part matches a search pattern and thus has a unique result color, use the part's result color. If a part does not match the search pattern, traverse up the assembly structure until a parent that matches the search pattern is found. Color the part with the result color of that parent.
	From the assembly down, include unmatched	Beginning at the assembly structure level you specify in Only at level, apply the result color of the node to itself and its children. Parts above the specified structure level are colored as if unmatched.
	From the assembly down, skip unmatched	Beginning at the assembly structure level you specify in Start at level, apply the result color of the node to itself and its children. If a node at the specified level does not match the search pattern, traverse down the assembly structure until a subassembly or part that matches the search pattern is found, then apply that result color. Parts above the specified structure level are colored as if unmatched.
Set the assembly level for searching assembly color matching.	From the Only at level or Start at level , list select an assembly structure level. Note: If you change any assembly level color setting, be sure to refresh the view by clicking Refresh.	
Save your assembly level coloring option as the default.	Select Set as default .	

Note:

- When you create visual reports and then review the results using **Results Legend**, parts and assemblies that you select in the **Results Legend** tree are highlighted in the **Assembly** view and in the Viewing window.
- When you select an assembly-based coloring option and specify an assembly structure level, in the **Results Legend** tree all parts and assemblies at the specified level are highlighted.

Assembly**Result Legend**

Save visual reports

You can create three different types of visual reports; quick, single rule, and multiple rules reports. You can also load and apply a saved visual report.

1. On the **Visual Report** toolbar, click any one of the following options:

- **Quick Report**

- **Single Rule Report**

- **Multiple Rules Report**

- **Apply Report**

2. Create the rules for your report and then click **Save As**.
3. In the **Save Visual Report File** dialog box, type the name of your report and then click **Save**.

The report is automatically saved to your local file system with a **.vpx** extension. It can be opened again when you **load a saved report**.

Edit visual reports

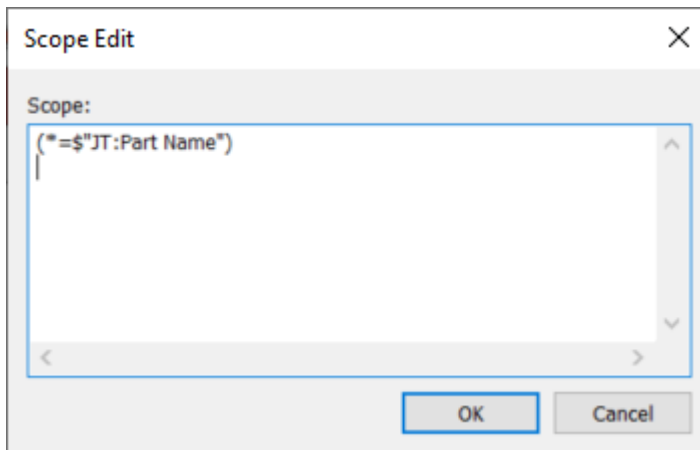
Use **Edit** to manually update and edit the report scope or to edit a saved visual report **.vpx** file. For example, you may want to expand your visual report to include parts that are designed by another engineer.

Note:

To edit visual reports you must set scoping to **Use local scoping**.

Edit scope rules

1. On the **Visual Report** toolbar, click **Visual Report** , and then click **Single Rule Report** .
2. Click **Edit** to open the **Scope Edit** dialog box.



3. Type your edits and click **OK**.

Edit a saved scope report (.vpx) file

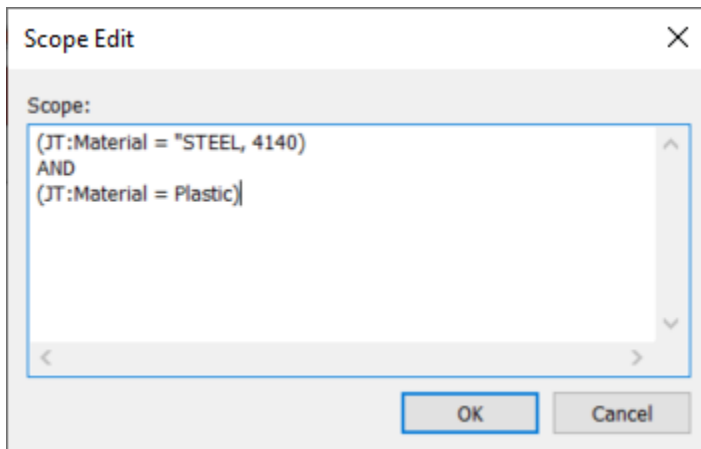
1. Click **Load** and in the **Load Visual Report File** dialog box that appears, and select the saved report you want to edit.
2. Click **Edit**.
3. Type your edits in the **Scope Edit** dialog box and click **OK**.

The results appear immediately in the Viewing window.

4. (Optional) Click **Save As** to save the updated visual report rule or rules.

Example

This example uses scope rules for a visual report to display two material values.



Reset visual report results

You can return to the previous visual appearance of your assembly.

On the **Visual Report** toolbar, click **Reset** .

If no visual report was applied to the assembly, a message appears prompting you to reset the appearance to the default materials.

If a visual report was applied to the assembly, the appearance is reset to the state before the report was applied.

Note:

If you click the **Visual Report** toolbar option  the data is restored to its original setting.

Specify preferences for visual reports

You can use visual report preferences to change default characteristics for how your visual report is displayed. You can override the default preferences by setting visual report characteristics at the time you create reports.

1. On the **3D Visual Report** toolbar, click **Preferences** .
2. On the **Appearance** tab, do any of the following:

In this section	Use this option	To
Matched Results Appearance	Color list	Specify the color of matched parts.
	Transparency	Specify the amount of color transparency of matched parts. Note: Move the slider to the right for greater transparency. Greater transparency can be useful when parts are small or when they are hidden within the model.
	Display mode	Specify how you want matched parts displayed. Note: You can display model shading, construction geometry, tessellation lines, and draw feature lines. You can select more than one mode.
Color Ramp	Edit	Specify color ramp options.
Unmatched Results Appearance	Use See-Thru transparency	Specify the transparency style to use when you display unmatched parts as transparent. Select the Use See-Thru transparency check box to turn this feature on and view unmatched parts easily, especially for complex visual reports. Clear this check box to view unmatched parts using standard transparency. Note: To change the transparency color and level, in the Viewing window, right-click and choose Preferences. Then set these values on the Display tab.
	Keep original visual appearance	Set the original appearance for unmatched parts.
	Color list	Specify the color of unmatched parts.
	Transparency	Specify the transparency level for unmatched parts.
	Display mode	Specify how you want unmatched parts displayed.

In this section	Use this option	To
		Note: You can display model shading, construction geometry, tessellation lines, and draw feature lines. You can select more than one mode.

3. In the **Legend** tab, do any of the following:

Use this option	To
Show Legend	Display the legend in the 3D view for the visual report.
Discrete Type	Specify the legend style. From the list, you can select band, scalar, bar chart, or pie chart.
Position	Specify where you want the legend to appear in the visual report.
Enable Paging	Display paging controls in the legend to move forward and backward through the results, when they have more items than the number specified for Items Per Page .
Legend Size	Specify the size of the legend.
Legend Text	Specify the title of the legend.
Insert Report Name	Specify if you want the visual report name to be automatically included in the legend display.
Legend Preferences	Specify Edge style and Font characteristics. Note: If the Visual Report legend does not display correctly using the default font for the target language, select a different font and font size until the legend displays as desired.
Show Unmatched for Multiple Rules report	Choose to display or hide unmatched parts and assemblies in the multiple rule visual report.

Use this option	To
	Note: This feature is only supported when you create a multiple rule report. By definition, the multiple rule visual report must consist of two or more rules.

Note:

The preferences you set in this section are applied to all reports. For example, the legend text is fixed.

4. In the **Reports** tab, do any of the following:

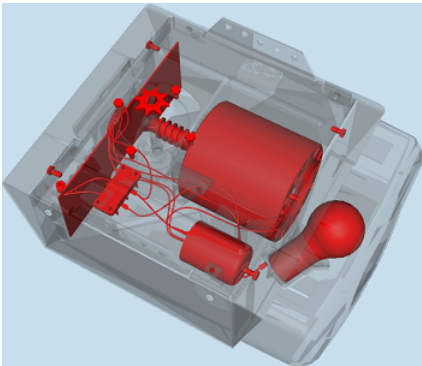
In this section	Do this
Scope	From the Default local scoping source list, choose one of the following: • TC Structure Lines • JT • JT and TC Structure Lines To make sure that attribute searches are case sensitive, select the Case sensitive for string comparison check box. To allow partially visible assemblies to be considered as visible, select the Consider partially visible assemblies as visible check box. If selected, when a visual report is generated, partially visible assemblies receive assembly-level coloring.
Results Legend	Select or clear the Show after applying a report check box to display the Results Legend dialog box.

In this section	Do this
	From the Selection style list, choose Material Color or Bounding Box Color , and select a color from the list.
Explore Non-Focus	Set the default Transparency level and the Display mode for parts that are not in the rule focus when you use Report Explore .

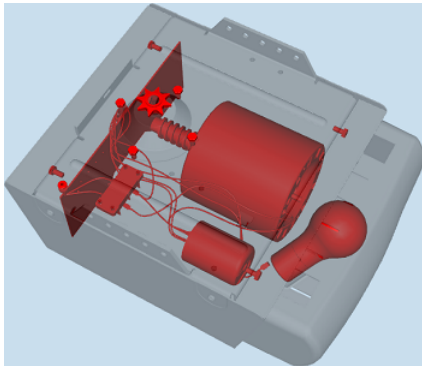
See-Thru transparency

When working with visual reports, you can choose to display unmatched parts as transparent. You can also set a visual reports **preference** to use standard transparency or **See-Thru transparency**.

- Standard transparency uses several layers of transparency.
- See-Thru transparency displays the model with one layer of transparency. Your view of key parts is less obstructed.



Visual report results
using standard transparency



Visual report results
using See-Thru transparency

Note:
To change the transparency color and level for both types, in the Viewing window, right-click and choose **Preferences**. Then set these values on the **Display** tab.

Working with path planning

Planning a path using Path Editor

Using Path Editor

Use the commands found in the **Path Editor** group to complete path planning tasks, such as path planning using **key positions** or user-aided fitting. **Path Editor** supports many of the same commands as **Path Planning**.

For **Swept Volume** generation, use the **Swept Volume** command from the **Swept Volume** group or from the **Paths** tab in the Project Workspace window.

Use **Path Planning** to access **Glide and Slide**.

Create a path in Path Editor using key positions

1. Select the target parts from the Viewing window or the Assembly view.
2. Choose **Actions**→**Path Editor**→**Create Path**.

The part manipulator and floating **Path Editor Toolbar** are displayed and the first key position is added at the position of the target part.

Note:

The floating **Path Editor Toolbar** is linked to the manipulator. To move the floating toolbar within the window, click the toolbar's gray outer box and drag to the desired position in the window. To turn off the floating toolbar, click **Path Editor Toolbar** .

3. Do one or more of the following.

To	Do this
Add a key position	Move the manipulator and selected parts to the desired position and click Add a new key position  .
Replace the selected key position with the manipulator's current position	Move the manipulator to the desired position and click Replace the selected key position with the manipulator's current position  .
Delete a key position	a. In the Viewing window, select the key position that you want to delete.

To	Do this
	b. Click Remove selected key position(s)  .
Detach the manipulator from the part	On the Path Editor Toolbar, click Detach  and then move the manipulator to the desired position. The manipulator's original position is indicated with by a dot in the Viewer. To reattach the manipulator to the part, click Attach .
Constrain parts	Click Open Path Planning Constraints  .
Resolve collisions at key positions	a. Select a key position from the list where target parts are in collision with background parts. (Collision appears in the Status field.) b. Click Move selected key position away from collision . c. In the Unstick dialog box, do one of the following. • Select Automatic to automatically resolve the collision. • Select Along direction to manually resolve the collision, and then do the following. A. (Optional) Select Check both directions. B. (Optional) Click Flip to invert the direction. C. In the X, Y, and Z boxes, type direction values. -or- Click Pick, and in the Viewing window, select a part feature. Note: • Select X, Y, or Z to update only a subset of coordinates. • For edge selections, the direction vector is formed by the edge selected. • For surface selections, the direction vector is formed by the vector normal to the selected surface, or the axis of a selected cylindrical surface.

To	Do this
	For point selections, the direction vector is formed by the point normal. d. Click Unstick.
Define the collision clearance setting	a. Click Path Editor Collision Settings . b. In the Path Editor Collision Settings dialog box, define the collision clearance value in the Default clearance field. Clearance appears in the Status column in the Advanced Editor dialog box for key positions that are under the defined value display. c. (Optional) Define specific collision sets. d. Click OK.
Launch user-aided fitting to define the path interactively	Allow user-aided fitting to assist you in defining the optimal path.

4. (Optional) To preview the path, click **Motion Path Preview** .

Use Path Planning to create a collision-free path

Use **Path Planning** to automatically determine a collision-free path between two key positions.

- Create two **key positions**.
- Choose **Actions**→**Path Editor**→**Launch Path Planning**.

The **Progress** dialog box displays as Visualization determines the collision-free path.

When a collision free path has been generated, the key positions display on the Viewing window.

- (Optional) On the **Progress** dialog box, click **Stop** to stop the path planning process.
- (Optional) Use the options in the **User Guided Path Generation** dialog box to use the magnetic guide to assist in the path planning by doing the following.

To	Do this
Activate or deactivate the magnetic guide	a. Click Activate/Deactivate Guide . The selected part turns blue. b. Guide the part through the desired path.
Move the selected part to the closest non-colliding position	Click Move Guide to closest valid position . This returns the part to the last known valid position (where there was no collision).

5. (Optional) Click **Motion Path Preview** to  review the planned path.

Define collision sets

Use collision sets to specify more precise clearance values between a subset of the target part and the defined obstacle part.

Collision set values affect user-aided fitting, path planning, the key position collision statuses in the **Advanced Editor** dialog box, and the Viewing window.

1. Select the target parts from the Viewing window or the Assembly view.
2. Choose **Actions**→**Path Editor**→**Create Path**.
3. Click **Path Editor Collision Settings** .

The **Path Editor Collision Settings** dialog box is displayed.

4. **Add**, **edit**, or **delete** a collision set, or create an excluded **parts** list.

Add a collision set

1. In the **Path Editor Collision Settings** dialog box, do the following.
 - a. (Optional) Enter a value in the **Default clearance** box to define the global clearance value.
This value is used by other path planning processes, but is *not* used by the collision sets.
 - b. Click **Add collision set** .

The **Collision Set Parts** dialog box is displayed.

- c. In the Viewing window, select a target part.
- d. In the **Collision Set Parts** dialog box, next to the **Target Parts** list, do one of the following:
 - Click **Add selected part(s) to target part list**  to add the selected target part to the list.
Repeat to add multiple parts to the list.
 - Click **Set all target parts**  to add all target parts to the list.
- e. In the Viewing window, select an obstacle part.
- f. In the **Collision Set Parts** dialog box, next to the **Obstacle Parts** list, do one of the following:
 - Click **Add selected part(s) to obstacle part list**  to add the selected obstacle part to the list.
Repeat to add multiple parts to the list.
 - Click **Set all obstacle parts**  to add all obstacle parts to the list.
2. (Optional) Select **Highlight target and obstacle parts in 3D view** to enable target and obstacle part coloring in the Viewing window.
3. In the **Collision Set Parts** dialog box, click **OK**.
4. In the **Path Editor Collision Settings** dialog box, do the following.
 - a. Click the default clearance value (0) on the new collision set and enter the desired clearance value for this set.
 - b. (Optional) Double-click the name of the new collision set and type a meaningful name.
5. (Optional) Create an excluded parts list for this set.
6. In the **Path Editor Collision Settings** dialog box, click **OK**.

Create an excluded parts list

Use the excluded parts list to identify parts to be ignored during path creation for all collision detection computations, such as clearance, minimal distance, key position statuses, and path planning.

1. In the Viewing window, select a part.

2. In the **Path Editor Collision Settings** dialog box, next to the **Excluded Parts** list, click **Add excluded part**  to add the selected part to the list.
3. (Optional) Add additional excluded parts.
4. Click **OK**.

Edit a collision set

1. In the **Path Editor Collision Settings** dialog box, select a collision set from the list.
2. Click **Edit selected collision set** .

The **Collision Set Parts** dialog box is displayed.

3. Add or delete target and obstacle parts from the part lists.
4. Click **OK**.

Delete a collision set or list parts

1. In the **Path Editor Collision Settings** dialog box, select a collision set or excluded part from the list.
-or-

In the **Collision Set Parts** dialog box, select a target or obstacle part from the list.

2. Click  next to the selected collision set or list part.
3. Click **OK**.

Constrain parts using the Path Editor

You can constrain the translation and rotation of target parts.

Constraint appears in the **Status** column in the **Advanced Editor** dialog box for key positions that violate the defined constraint values.

Example:

For example, you can specify to keep a part in the upright position while it is moving along the extraction path. You can define each axis as free, bounded between a minimum value and a maximum value, or locked to a value.

1. Define **key positions** for the target parts.
2. Choose **Actions→Path Editor→Open Path Planning Constraints**.
3. In the **Path Planning Constraints** dialog box, do any of the following:

To	Do this
Constrain the translation movement of the target parts	- Select one or more axes (X, Y, and Z). - Define the Low and High limits for each axis. The values are relative to the first key position. The verbiage at the bottom of the dialog indicates whether the translation values are valid or invalid.
Constrain the rotation movement of the target parts	- Select one or more rotational axes (Rx, Ry, and Rz). - Define the Low and High limits for each rotational axis. The values are relative to the first key position. The verbiage at the bottom of the dialog indicates whether the rotation values are valid or invalid. (Optional) Click Simulate Rx, Ry, or Rz to preview the rotation movement of the target parts.
Lock specific translation or rotation axes	Select the checkbox beneath the lock icon that corresponds to the axis that you want to lock. The High and Low values are set to 0 for the locked axis. If Display limits in View is selected, locking the constraints will affect how the 3D walls display. For example, if you lock the X axis, only the start position 3D wall displays.
See a 3D graphical representation (3D walls) of the path planning area in the viewing window	Select Display limits in view. The defined path limit is bounded by 3D walls. Tip: • Manipulate the 3D walls in the viewer to easily adjust the constraint values. • The manipulator may become obstructed by the 3D walls. If this happens, clear the checkbox to move the manipulator.

- Click **OK**.

Edit an existing path using Path Editor

Use the **Advanced Editor** dialog box to modify an existing path. The value in the **Status** field indicates if there is a collision between a key position and another part.

- Select a key position on the path that you want to edit.
- Choose **Actions**→**Path Editor**→**Advanced Editor**.

Use the up and down arrows to move between key positions within the **Key Position** list in the **Advanced Editor** dialog box.

- Do one or more of the following.

To	Do this
Add a key position	- Select a key position from the list. - Click . The new key position is added below.
Replace the selected key position with the manipulator's current position	- In the Viewing window, move the manipulator to the desired position. - Select the key position from the list that you want to replace. - Click Replace the selected key position with the manipulator's current position .
Delete a key position	- Select the key position from the list that you want to delete. Use the Shift or Ctrl key to select several key positions. - Click Remove selected key position(s) .
Resolve collisions at key positions	- Select a key position from the list where target parts are in collision with background parts. (Collision appears in the Status field.) - Click Move selected key position away from collision . - In the Unstick dialog box, do one of the following. - Select Automatic to automatically resolve the collision.

To	Do this
	- Select Along direction to manually resolve the collision, and then do the following. (Optional) Select Check both directions. (Optional) Click Flip to invert the direction. - In the X, Y, and Z boxes, type direction values. -or- Click Pick, and in the Viewing window, select a part feature. Note: - Select X, Y, or Z to update only a subset of coordinates. - For edge selections, the direction vector is formed by the edge selected. - For surface selections, the direction vector is formed by the vector normal to the selected surface, or the axis of a selected cylindrical surface. - For point selections, the direction vector is formed by the point normal. d. Click Unstick.
Manually adjust the path by changing a key position's type	Right-click the key position and select from the following type values: None, Start, Intermediate, Goal. An extraction path must consist of at least two key positions, a start position and an end position, for the target parts. You also can define intermediate positions to constrain the path direction. If None is selected, the key position is not used in the path.

- Click **Close**.

User-aided fitting in Path Editor

User-aided fitting mode provides assistance in creating a smooth, optimized, collision-free path.

- Select a part and create a **key position**.

2. Choose **Actions→Path Editor→User Aided Fitting**.
3. Continue to move the manipulator to the desired location. The user-aided fitting algorithm defines the best path, defining key positions as the path is defined.

A polyline shows the glide path while you move the part. The **Show Trajectory Path** preference controls the display of the glide path.

Note:

In user-aided fitting mode, when you move the target parts into contact with the background parts, the contacts are highlighted per the Collision Display settings.

4. Once the initial path is complete, choose **Actions→Path Editor→User Aided Fitting**.

The final position is automatically added as a key position.

Specify path editor preferences

You can customize the path editor default settings.

1. Choose **Actions→Path Editor→Path Editor Preferences**.
2. In the **Preferences** dialog box, choose any of the following:

On this page	Use this option	To do this
General	Automatically Display Path Editor Toolbar	Shows the floating Path Editor Toolbar in the Viewing window when path editing begins.
	Automatically Display Advanced Editor	Shows the Advanced Editor dialog box in the Viewing window when path editing.
	Update graphics	Shows a "ghost" of the target parts during path planning, indicating the last collision-free position found by the algorithm.
	Show Path Trajectory	Shows the trajectory path in the 3D Viewing window during the definition of the simulation path.
Collision Display	Show Parts in Collision	Shows which parts collide with other parts along the path. Parts in violation are shown in the selected Highlight Color and Transparency .
	Show Minimum Distance Line	Single line indicating the minimum distance between target and background parts. Distance lines are shown in the selected Line Color and Line Thickness .

On this page	Use this option	To do this
	Show Distance	Displays the distance between parts based on the value defined in Model Units .
	Show Intersection Lines	Displays intersecting lines along the path. Lines that intersect are displayed in the selected Line Color and Line Thickness .

Planning a path using Path Planning

Selecting a path planning method

Use path planning to identify extraction paths for parts or assemblies that need to be removed from models for maintenance or replacement. Path planning is supported in both standard desktop and immersive environments.

Note:

Path Planning is turned off on the Ribbon bar and in the Actions Menu by default. To turn on Path Planning, add the Windows environment variable TCVIS_ENABLE_PATHPLANNING, and set it to True.

When you generate a new path, you select one of the following three methods and **specify the target and background parts**.

- **Key positions**

Defines key positions to guide the general direction of the target parts as they move along the extraction path. Key positions are points within the geometric space occupied by the model that target parts must pass through. Lifecycle Visualization calculates an extraction path to pass through the key positions.

Use this method when the path is relatively clear and simple.

- **User-Aided Fitting**

Calculates a collision-free path interactively while you move the target assemblies. It combines the computing power of Lifecycle Visualization with your knowledge of the best path to take.

Use this method when you need to move the path through one or two areas of tight tolerances.

- **Glide and Slide**

Calculates a collision-free path interactively that pivots around collisions with background parts.

Note:

- This requires JT files with XT B-Rep data. XT B-Rep with tolerant geometry is not supported.
- If mass properties are not available in the part, a constant mass is assumed.
- Constraints between components are not considered.
- ALSG and tessellated data are not supported.

Use this method when you need a path through a complex assembly with tight spaces.

Specify target and background parts

Before you can generate a part extraction path, you must first specify target and background parts. The parts that you want to move or extract from your model are the target parts. The parts that must be avoided are the background parts.

1. Choose **Actions**→**Path Planning**→**Generate New Path**.

The **New Path** dialog box is displayed.

2. Click a button in the **Methods** section to specify how you want to generate a new path:

Button	Does this
Key Positions	Specifies key positions to guide the general direction of the target parts as they move along the extraction path.
User-Aided Fitting	Enables the interactive creation of a collision-free path that is analyzed and refined in real-time while you are moving the target parts or assembly.
Glide and Slide	Simulates the motion of parts by taking account of the interaction of parts with other parts in the assembly as they glide-and-slide against other parts.

3. To specify target parts, do either of the following:

To specify target parts	Do this
Based on part selection	a. In the Viewing window or the assembly tree, select one or more parts. b. In the Target Parts list, select <Create from selected>.
Based on part visibility	a. In the Viewing window or the assembly tree, display only the parts that you want to specify as target parts.

To specify target parts	Do this
	b. In the Target Parts list, select <Create from visible> .

4. In the **Alternate Hierarchy Name** dialog box, type a name for the target parts and click **OK**.

The target parts appear as an **alternate hierarchy** in the assembly tree.

5. To specify background parts, do either of the following:

To specify background parts	Do this
Based on part selection	a. In the Viewing window or the assembly tree, select one or more parts. b. In the Target Parts list, select <Create from selected>.
Based on part visibility	a. In the Viewing window or the assembly tree, display only the parts that you want to specify as background parts. If target parts are also visible, they are not included with the background parts. b. In the Target Parts list, select <Create from visible (exclude target parts)>.

6. In the **Alternate Hierarchy Name** dialog box, type a name for the background parts and click **OK**.

The background parts appear as an **alternate hierarchy** in the assembly tree.

7. (Optional) To see which parts are now defined as target parts or background parts, click **Select** in the **Target Parts** or **Background Parts** sections, respectively.

Visible parts are selected in the Viewing window.

8. You cannot define the same parts as both target and background parts. If you have created target and background parts that consist of one or more of the same parts, click **Select** in the **Part Options** section to identify the duplicate parts in the Viewing window.

Tip:

- If you add or remove target or background parts after defining key positions, parts in key positions may collide.

- After specifying target and background parts, the next step in the path planning process is to define either key positions or the path. On the bottom right of the **New Path** dialog box, click **Next** to change to the next tab.

Planning a path through key positions

Overview of key positions

You must specify key positions to guide the general direction of the target parts as they move along the extraction path. Key positions are points within the geometric space occupied by the model that target parts must pass through during the calculation of the extraction path.

A part extraction path must consist of at least two key positions, a start position and an end position for the target parts. You also can define intermediate positions to constrain the path direction or to generate the path faster.

Target parts in key positions cannot collide with background parts. When you generate an extraction path, part collisions and constraint failures are detected and identified in the **Key Positions** list.

Note:

You can choose to ignore collisions in the start and end positions.

Define key positions

Define **key positions** to guide the general direction of the target parts as they move along the extraction path.

1. Specify **target and background parts**.
2. In the **New Path** dialog box, click the **Key Positions** tab.
3. Do one or more of the following to create and manipulate key positions:

To	Do this
Select the target parts	Click Select Parts .
Constrain parts	Click Constrain .
Add a key position using part manipulation mode	a. Select Part Manipulation Mode. b. Move the target parts to a new position. c. Click Add.

To	Do this
Add a key position using the Part Transformation dialog box	- Select Part Transformation. - In the Part Transformation dialog box, translate or rotate the target parts until they are in a new position, and click OK. - Click Add.
Add a key position using part manipulators	- Select Add Manipulator. Move the target parts to a new position. - Click Add.
Replace a key position	- Translate the target parts until they are in a new position. - In the Key Positions list, select the key position that you want to replace. - Click Replace.
Delete a key position	- In the Key Positions list, select the key position that you want to delete. - Click Remove.
Reorder key positions	- In the Key Positions list, select the key position that you want to move. - Click Move Up or Move Down. The key position moves up or down in the list.
Navigate the Key Positions list	- In the Key Positions list, select a key position. - Click Previous or Next. The previous or next key position is selected.
Move the target parts to a key position	In the Key Positions list, double-click a key position.
Resolve collisions at key positions	- In the Key Positions list, select a key position where target parts are in collision with background parts. - Click Unstick.

To	Do this
	Note: You must generate a path for Unstick to become available.
Ignore collisions in the start position when you generate a path	Select the Ignore collisions in first segment check box.
Ignore collisions in the end position when you generate a path	Select the Ignore collisions in last segment check box.
Open a VFM motion file that contains key positions	Click Open, and open the VFM file. Key positions in the file appear in the Key Positions list.
Save key positions as a VFM motion file	Click Save , and save the file.

Tip:

After defining key positions, you can **constrain** your target parts or **generate** the extraction path. On the bottom right of the **New Path** dialog box, click **Next** to change to the **Path Generation** tab.

Constrain parts

You can constrain the translation and rotation of target parts.

Example:

For example, you can specify to keep a part in the upright position while it is moving along the extraction path. You can define each axis as free, bounded between a minimum value and a maximum value, or locked to a value.

1. Specify **target and background parts**.
2. Define **key positions** for the target parts.
3. In the **New Path** dialog box, click the **Key Positions** tab.
4. In the **Target Parts** section, click **Constrain**.
5. In the **Constrain Target Parts** dialog box, do any of the following:

To	Do this
Specify a reference position to which all constraints are relative	Click Capture .
View a specified reference position	Click Show .
Use the global origin to describe the translation constraints	Select the Translation relative to global center check box.
Constrain the translation movement of the target parts	a. For each axis (X, Y, and Z) choose one of the following: • Free • Bounded • Locked b. (For Bounded) Type minimum and maximum values. c. (For Locked) Type the degree.
Constrain the rotation movement of the target parts	a. For each axis (X, Y, and Z) choose one of the following: • Free • Bounded • Locked b. (For Bounded) Type minimum and maximum values. c. (For Locked) Type the degree.

Generate extraction paths using key positions

You can generate extraction paths for parts or assemblies that need to be removed from models for maintenance or replacement.

1. Specify **target and background parts**.
2. Define **key positions** for the target parts.
3. (Optional) **Constrain** the target parts.
4. In the **New Path** dialog box, click the **Path Generation** tab.

5. In the **Static Clearance** box, type a value for the minimum distance that you want to allow between target parts and background parts in key positions.
6. (Optional) Select the **Create swept volume** check box.

Note:

Click **Configure** to **adjust the chordal tolerance** used to generate the swept volume.

7. (Optional) Select **Guide Path**, click **Browse**, and then open a VFM file.
8. Click **Generate Path**.

The part extraction path is generated. If there are collisions or constraint failures between target parts and background parts, the **Path Generation Error** dialog box appears.

9. (If an error occurs) In the **Path Generation Error** dialog box, click one of the following:

- **Check Collision Report**

The **Collision Report** page appears. You can use this page to examine background parts in collision with target parts.

- **Edit Key Position**

The **Key Positions** page appears. Collisions and constraint failures are identified in the **Key Positions** list. Use the **Unstick** dialog box or manually transform the target parts to **resolve the errors**.

10. (Optional) Click **Play** to display the **Motion Playback** dialog box and **view the motion** in the Viewing window.
11. (Optional) Click **Save**, and save the generated solution path as a VFM motion file.

Specify a time limit for path planning calculations

If the application cannot identify a path in your model, the path planning calculation will continue indefinitely, unless you use the path planning preferences to stop the calculation after a specified duration.

1. Choose **Actions**→**Path Planning**→**Preferences**.
2. In the **Preferences** dialog box on the **General** page, select the **Stop calculation if solution is not found within time limit** check box.

3. In the **Time Limit** box, specify how many minutes you want the application to search for a path before stopping.
4. If you want to display a confirmation dialog when a solution is not found within the specified time limit, select the **Show Stop Calculation Confirmation dialogs** check box.

When you run a path planning calculation, if the application cannot find a path in the specified time, a dialog box is displayed where you can choose to continue or stop the calculation.

Resolve collisions

Overview of resolving collisions

When you attempt to generate a part extraction path, collisions between parts at key positions are identified in the **Key Positions** list. Use the **Unstick** dialog box to find a collision free position for the target parts. You can specify to move the target parts **along a specified direction** or **within a bounded region**.

Note:

- Unstick ignores **constraints** defined for target parts.
- You can choose to **ignore collisions** in the start and end positions.

Tip:

Use the **path planning preferences** to specify for the application to resolve collisions automatically while generating the solution path.

Resolve collisions along a direction

1. In the **New Path** dialog box, click the **Key Positions** tab.
2. Select a key position where target parts are in collision with background parts, and click **Unstick**.
3. In the **Unstick** dialog box, choose **Along Direction**.
4. (Optional) Select **Check Both Direction**.
5. In the **X**, **Y**, and **Z** boxes, type direction values.

-or-

Click **Pick**, and in the Viewing window, **select a part feature**.

Note:

- Select **X**, **Y**, or **Z** to update only a subset of coordinates.
- For edge selections, the direction vector is formed by the edge selected.
- For surface selections, the direction vector is formed by the vector normal to the selected surface, or the axis of a selected cylindrical surface.
- For point selections, the direction vector is formed by the point normal.

6. (Optional) Click **Flip** to invert the direction.

7. Click **Unstick**.

Resolve collisions within a specified region

1. In the **New Path** dialog box, click the **Key Positions** tab.
2. Select a key position where target parts are in collision with background parts, and click **Unstick**.
3. In the **Unstick** dialog box, choose **Within Bounded Region**.
4. In the **Minimum Radius**, type a value.
5. In the **Maximum Radius**, type a value.
6. In the **Scale Factor**, type a value.
7. Click **Unstick**.

User-aided fitting in path planning

Define the path interactively

Once you have selected either the User-Aided Fitting or Glide and Slide method and specified the target and background parts, you can define the path of the target parts.

1. Click the **Select** button to highlight the target parts you selected on the **General** tab.
2. In the **New Path** dialog box, click the **Define Path** tab and select one of the following check boxes:

- **Part Manipulation Mode**

Translate parts by dragging them around the Viewing window.

- **Part Transformation**

Translate parts by setting values in the **Part Transformation** dialog box.

- **Add Manipulator**

Translate parts by moving the manipulator around the Viewing window.

3. Click the **Start** button to begin the path and move the target part.

The initial position is automatically added as a key position.

A polyline shows the glide path while you move the part. The **Show Trajectory Path** preference controls the display of the glide path.

Note:

In user-aided fitting mode, when you move the target parts into contact with the background parts, the target parts become transparent. You can release the target parts at this point by releasing the mouse button or by halting movement of the 3DConnexion device. User-aided fitting automatically moves the target parts away from the collision state after you release them, and they return to their original color.

4. Once the initial path is complete, click the **Stop** button.

The final position is automatically added as a key position.

5. (Optional) To save the path and key positions to VFM files click **Save Path**, enter file names, and click **OK**.

Modify an existing user-aided fitting path

1. If the path is already saved, in the **New Path** dialog box, click the **Define Path** tab, and click **Load Path**.
2. Browse for the key positions and guide path file names and click **OK**.
3. Click the **Start** button to resume recording the path.
4. Do one or more of the following to edit the glide path:

To	Do this
Move the target parts to another position	Right-click the position and choose View Target Position .
Convert an intermediate position to a key position	Right-click the position and choose Promote to Key Position .
Convert a key position to an intermediate position	Right-click the position and choose Demote to Intermediate Position .
Remove a key position	Right-click the position and choose Remove .
Perform other actions to key positions.	Open the New Path dialog box, click the Define Path tab, and use the Add, Remove, Auto, Unstick, or Clear Path buttons .

Define key positions for an interactive path

1. In the **New Path** dialog box, click the **Define Path** tab.
2. Do one or more of the following to create and manipulate key positions:

To	Do this
Select the target parts	Click the Select button.
Add a key position using part manipulation mode	a. Select Part Manipulation Mode. b. Move the target parts to a new position. c. Click Add.
Add a key position using the Part Transformation dialog box	a. Select Part Transformation. b. In the Part Transformation dialog box, translate or rotate the target parts until they are in a new position, and click OK. c. Click Add.
Add a key position using part manipulators	a. Select Add Manipulator. b. Move the target parts to a new position. c. Click Add.
Automatically generate key positions based on the curvature of the path	Click Auto .

To	Do this
Delete the last key position	In the Key Positions list, click Remove .
Delete all key positions and clear the path	Click Clear Path .
Navigate through the intermediate positions in the path	a. In the Key Positions list, select a key position. b. Click Previous << , Next >> , Beginning < , or End > .
Move the target parts to a key position	In the Key Positions list, double-click a key position.
Resolve collisions at key positions	a. In the Key Positions list, select a key position where target parts are in collision with background parts. b. Click Unstick . Note: You must generate a path for Unstick to become available.
Open VFM motion files that contains key positions and a guide path	Click Load Path , and select a key positions VFM file and a guide path VFM file. Key positions in the file appear in the Key Positions list.
Save key positions and guide path as VFM motion files	Click Save Path , and save the file.

Generate extraction paths

You can generate extraction paths for parts or assemblies that need to be removed from models for maintenance or replacement after defining paths with user-aided fitting or glide and slide.

1. Specify **target and background parts**.
2. In the **New Path** dialog box, click the **Path Generation** tab.
3. In the **Static Clearance** box, type a value for the minimum distance that you want to allow between target parts and background parts in key positions.
4. (Optional) Select the **Create swept volume** check box.

Note:

Click **Configure** to adjust the chordal tolerance used to generate the swept volume.

5. (Optional) Select **Guide Path**, click **Browse**, and then open a VFM file.
6. Click **Generate Path**.

The part extraction path is generated. If there are collisions or constraint failures between target parts and background parts, the **Path Generation Error** dialog box appears.

7. (If an error occurs) In the **Path Generation Error** dialog box, click one of the following:

- **Check Collision Report**

The **Collision Report** page appears. You can use this page to examine background parts in collision with target parts.

- **Edit Key Position**

The **Key Positions** page appears. Collisions and constraint failures are identified in the **Key Positions** list. Use the **Unstick** dialog box or manually transform the target parts to **resolve the errors**.

8. (Optional) Click **Play** to display the **Motion Playback** dialog box and **view the motion** in the Viewing window.
9. (Optional) Click **Save**, and save the generated solution path as a VFM motion file.

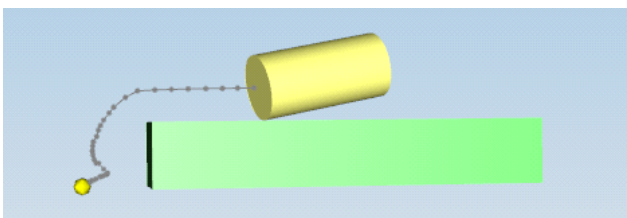
Glide and slide parts through a path

Create a path dynamically with glide and slide

You can select glide and slide to interactively create a dynamically-defined collision-free path for target assembly components that is analyzed and refined in real-time while you move the target parts. While you move the parts, if they contacts any other parts, they may rotate around a pivot point and glide-and-slide through the surface of other parts in contact.

Note:

Glide and slide is only available when using Path Planning. Path Planning is turned off on the Ribbon bar and in the Actions Menu by default. To turn on Path Planning, add the Windows environment variable TCVIS_ENABLE_PATHPLANNING, and set it to True.

Step	Example
Move the target part	
Target part contacts other parts	
Target part glides and slides over the surface of other parts	

Note:

In glide and slide mode, when you move the target parts into contact with the background parts, it may take some time to resolve the gliding path. The time required depends on the complexity of the geometry—data that contains tolerant geometry may take a longer time. If glide and slide cannot resolve the path and abort the operation, it pushes the target parts away from the collision state, and the path color turns red. The path turns back to its original black color when you manually move the target parts away from the collision state.

Define the path interactively

Once you have selected either the User-Aided Fitting or Glide and Slide method and specified the target and background parts, you can define the path of the target parts.

1. Click the **Select** button to highlight the target parts you selected on the **General** tab.
2. In the **New Path** dialog box, click the **Define Path** tab and select one of the following check boxes:
 - **Part Manipulation Mode**
Translate parts by dragging them around the Viewing window.
 - **Part Transformation**

Translate parts by setting values in the **Part Transformation** dialog box.

- **Add Manipulator**

Translate parts by moving the manipulator around the Viewing window.

3. Click the **Start** button to begin the path and move the target part.

The initial position is automatically added as a key position.

A polyline shows the glide path while you move the part. The **Show Trajectory Path** preference controls the display of the glide path.

Note:

In user-aided fitting mode, when you move the target parts into contact with the background parts, the target parts become transparent. You can release the target parts at this point by releasing the mouse button or by halting movement of the 3DConnexion device. User-aided fitting automatically moves the target parts away from the collision state after you release them, and they return to their original color.

4. Once the initial path is complete, click the **Stop** button.

The final position is automatically added as a key position.

5. (Optional) To save the path and key positions to VFM files click **Save Path**, enter file names, and click **OK**.

Modify an existing glide-and-slide path

1. If the path is already saved, in the **New Path** dialog box, click the **Define Path** tab, and click **Load Path**.
2. Browse for the key positions and guide path file names and click **OK**.
3. Click the **Start** button to resume recording the path.
4. Do one or more of the following to edit the glide path:

To	Do this
Move the target parts to another position	Right-click the position and choose View Target Position .
Convert an intermediate position to a key position	Right-click the position and choose Promote to Key Position .

To	Do this
Convert a key position to an intermediate position	Right-click the position and choose Demote to Intermediate Position .
Remove a key position	Right-click the position and choose Remove .
Perform other actions to key positions.	Open the New Path dialog box, click the Define Path tab, and use the Add, Remove, Auto, Unstick, or Clear Path buttons .

Define key positions for an interactive path

1. In the **New Path** dialog box, click the **Define Path** tab.
2. Do one or more of the following to create and manipulate key positions:

To	Do this
Select the target parts	Click the Select button.
Add a key position using part manipulation mode	a. Select Part Manipulation Mode. b. Move the target parts to a new position. c. Click Add.
Add a key position using the Part Transformation dialog box	a. Select Part Transformation. b. In the Part Transformation dialog box, translate or rotate the target parts until they are in a new position, and click OK. c. Click Add.
Add a key position using part manipulators	a. Select Add Manipulator. b. Move the target parts to a new position. c. Click Add.
Automatically generate key positions based on the curvature of the path	Click Auto .
Delete the last key position	In the Key Positions list, click Remove .
Delete all key positions and clear the path	Click Clear Path .

To	Do this
Navigate through the intermediate positions in the path	- In the Key Positions list, select a key position. - Click Previous <<, Next >>, Beginning <, or End > .
Move the target parts to a key position	In the Key Positions list, double-click a key position.
Resolve collisions at key positions	- In the Key Positions list, select a key position where target parts are in collision with background parts. - Click Unstick. Note: You must generate a path for Unstick to become available.
Open VFM motion files that contains key positions and a guide path	Click Load Path, and select a key positions VFM file and a guide path VFM file. Key positions in the file appear in the Key Positions list.
Save key positions and guide path as VFM motion files	Click Save Path , and save the file.

Generate extraction paths

You can generate extraction paths for parts or assemblies that need to be removed from models for maintenance or replacement after defining paths with user-aided fitting or glide and slide.

- Specify **target and background parts**.
- In the **New Path** dialog box, click the **Path Generation** tab.
- In the **Static Clearance** box, type a value for the minimum distance that you want to allow between target parts and background parts in key positions.
- (Optional) Select the **Create swept volume** check box.

Note:

Click **Configure** to adjust the chordal tolerance used to generate the swept volume.

- (Optional) Select **Guide Path**, click **Browse**, and then open a VFM file.
- Click **Generate Path**.

The part extraction path is generated. If there are collisions or constraint failures between target parts and background parts, the **Path Generation Error** dialog box appears.

7. (If an error occurs) In the **Path Generation Error** dialog box, click one of the following:

- **Check Collision Report**

The **Collision Report** page appears. You can use this page to examine background parts in collision with target parts.

- **Edit Key Position**

The **Key Positions** page appears. Collisions and constraint failures are identified in the **Key Positions** list. Use the **Unstick** dialog box or manually transform the target parts to **resolve the errors**.

8. (Optional) Click **Play** to display the **Motion Playback** dialog box and **view the motion** in the Viewing window.
9. (Optional) Click **Save**, and save the generated solution path as a VFM motion file.

Using guide paths

Overview of using guide paths

You can save the extraction path as a VFM motion file for use as a guide path in different work sessions. Guide paths enable you to generate subsequent path revisions faster.

Note:

For the current work session, the extraction path is retained in memory and automatically used as a guide for path optimization.

Saving guide paths

1. Generate an extraction path.
2. In the **New Path** dialog box, click the **Path Generation** tab.
3. Click **Save**, and save the path as a VFM file.

Load guide paths

1. In the **New Path** dialog box, click the **Path Generation** tab.
2. Click **Browse**, and open a VFM file that contains an extraction path.

Note:

The **Browse** button is available only when you use key positions as the input method. If you use user-aided fitting or glide-and-slide, you can load the guide path by clicking the **Load Path** button in the **Define Path** page.

Validate extraction paths

You can verify that a previously generated path is still valid with new part revisions.

1. Choose **Actions→Motion→Open Motion File**.
 2. In the **Select Motion Files** dialog box, select the VFM file that contains the extraction path you want to validate, and click **Open**.
 3. Choose **Tools→Clearance**.
 4. Choose **Clearance→Set Clearance**.
 5. In the **Set Clearance** dialog box, type a clearance value based upon one of the following conditions:
 - If the effective or final clearance value that was used to generate the extraction path is positive, type the value, and click **OK**.
 - If the effective or final clearance value is negative, type the static clearance value that was used to generate the path, and click **OK**. Then do the following:
 - a. Choose **Clearance→Preferences→Calculator Settings**.
 - b. In the **Clearance Calculator Settings** dialog box, select **Distinguish between Contact and Penetrations**, and click **OK**.
 - c. On the **Clearance** toolbar, click **Dynamic Clearance** .
 - d. In the **Dynamic Clearance** dialog box, select **Generate Results List**, and click **Enable**.
- Tip:**

When you play the motion, view the **General Clearance Results** list to ensure the penetration value is not larger than the final clearance value.
6. In the **Motion Playback** dialog box, **play the motion file**.

If no collisions are detected, the extraction path is valid with the newly revised parts. If collisions are detected, you may need to update the extraction path to account for the revised parts.

View collision reports

The collision report is generated if the extraction path fails to generate. The collision report displays a list of background parts in collision with target parts, along with the percentage of time the parts are in collision. You can zoom to each background part in the list to examine the parts in conflict.

1. In the **New Path** dialog box, click the **Collision Report** tab.
2. Do any of the following:

To	Do this
Select background parts in the Viewing window	In the list of background parts in collision, select one or more parts.
Move backward or forward through the collision list	In the list of background parts in collision, select a part, and click Prev or Next .
Zoom to a background part	In the list of background parts in collision, select a part, and click Fly to Parts .

Save path planning information

You can preserve alternate hierarchies used for target and background parts with session files. You also can save key positions and extraction paths as VFM motion files.

1. Specify **target and background parts**.
2. Define the **key positions** or **guide path** for the target parts.
3. Generate an **extraction path**.
4. Do any of the following:

To	Do this
Save target and background parts	Save the current work session as a session file.
Save key positions	One of the following: • If using the Key Positions method, click the Key Positions page of the New Path dialog box, click Save, and save the key positions as a VFM motion file.

To	Do this
	If using the User-Aided Fitting or Glide and Slide method, click the Define Path page of the New Path dialog box, click Save Path, and save the key positions as a VFM motion file.
Save guide path	If using the User-Aided Fitting or Glide and Slide method, click the Define Path page of the New Path dialog box, click Save Path , and save the guide path as a VFM motion file.
Save the extraction path	On the Path Generation page of the New Path dialog box, click Save , and save the extraction path as a VFM motion file.

Specify path planning preferences

You can customize the path planning default settings.

1. Choose **Actions→Path Planning→Preferences**.
2. In the **Preferences** dialog box, choose any of the following:

On this page	Use this option	To do this
General	Stop calculation if solution is not found within time limit	Stop the calculation after a specified amount of time. In the minutes box, type a value.
	Update graphics	Move the target parts in the 3D Viewing window while generating a path. Turning on this preference affects performance.
	Run optimization	Immediately after a solution path is found, automatically run path optimization to find the shortest possible path. In the number of iterations box, type a value for how many times the application should attempt to find a shorter path. Tip: If an extraction path uses a lot of geometric space and the target parts move very far outside the geometry, increase the number of iterations run by the optimization to reduce the extra geometric space.
	Automatically unstick key positions	Automatically resolve part collisions for key positions.

On this page	Use this option	To do this
	Automatically load target and background parts	Automatically load the first alternate hierarchy defined in the document for the target parts, and the second alternate hierarchy defined in the document for the background parts.
	Default Transformation Mode	Define the default transformation mode to be used during path planning. • Part Manipulation Mode — translate parts by dragging them around the Viewing window. • Part Transformation — translate parts along the axes or radially. • Add Manipulator — translate parts by moving the manipulator around the Viewing window.
	Show Path Trajectory	Shows the trajectory path in the 3D Viewing window during the definition of the simulation path.
	Show Stop Calculation Confirmation dialogs	Display a confirmation dialog box when calculations are canceled.
	Show Required Input dialogs	Display a warning if the input values required for the New Path dialog box are incomplete.
Collision Display	Show Parts in Violation	Shows which parts collide with other parts along the path. Parts in violation are shown in the selected Highlight Color and Transparency .
	Show Minimum Distance	Single line indicating the minimum distance between target and background parts.
	Line Ends	Define the line Style and Size for both the start and end of the Show Minimum Distance line.
	Show Distance	Displays the distance between parts based on the value defined in Model Units .
	Show Intersection Lines	Displays intersecting lines along the path. Lines that intersect are displayed in the selected Line Color and Line Thickness .
Glide and Slide	Glide And Slide Step	Define the step that glide and slide uses to compute the collision.

On this page	Use this option	To do this
		Note: - If you select a small step, glide and slide performance may decrease because of the time needed to compute all of the steps. - If you select a large step, performance increases, but the glide and slide effect may not be accurate.
	Snap Distance	Automatically captures an intermediate frame position while performing a glide-and-slide operation if the <i>linear</i> displacement between the previous captured frame position and current position is greater than or equal to the specified snap distance. The intermediate positions are positions captured between two successive simulation key frames.
	Snap Angle	Automatically captures an intermediate frame position while performing a glide-and-slide operation if the <i>angular</i> displacement between the previous captured frame position and current position is greater than or equal to the specified snap angle. The intermediate positions are positions captured between two successive simulation key frames.

Performance tips for path planning

The following tips, in order of priority, can help you to minimize the time it takes for the application to find a path planning solution:

1. Ensure the first key position is the most constrained position. The application calculates an extraction path faster than an assembly path, since there are fewer possible solutions when extracting a part from a small space.
2. Ensure the background parts consist only of relevant parts. This reduces the collision checking time required for the calculation. For example, if you want to extract something from the engine of a car, do not include the parts at the rear of the car as background parts. Always include only the parts that have a possibility of collision with the target part.
3. Turn off **Update graphics** in the path planning preferences. This eliminates the extra time it takes for the application to update the graphics while finding a solution.

4. Turn off **Run optimization** in the path planning preferences. When enabled, this option runs after a solution path is found, shortening the path when possible.

Creating swept volumes

Overview of creating swept volumes

You can create a graphical representation of the volume and path of a part as it moves.

Example:

You create an extraction path for an assembly that must be removed periodically for maintenance. You then create a swept volume of the assembly's extraction path, and insert the resulting .jt file into your document. As you design your model, you can see the swept volume and avoid placing parts in positions that interfere with the assembly removal.

Note:

To generate swept volumes, your machine must have a graphics card with a 24-bit depth buffer.

Creating swept volumes involves the following:

- Swept volume performance considerations
- Generate swept volumes from moving parts
- Edit the input path of swept volumes
- Generate swept volumes from stationary parts
- Specify swept volume preferences
- Save swept volumes

Swept volume performance considerations

The performance of swept volume creation is influenced by your graphics hardware, the presence of multiple CPUs or cores, physical memory, and the specified chordal tolerance.

Graphics hardware

To generate swept volumes, your machine must have a graphics card with a 24-bit depth buffer. The application utilizes the parallel processing power of modern graphics adapters to create the swept volume envelope. A faster graphics adapter results in quicker swept volume generation.

Multiple CPUs or cores

Machines possessing multiple CPUs or cores deliver additional modest performance improvements, as the application does make use of them to generate swept volumes. However, because the graphics adapter performs the majority of the processing, the benefit of having more than two cores decreases rapidly.

Physical memory

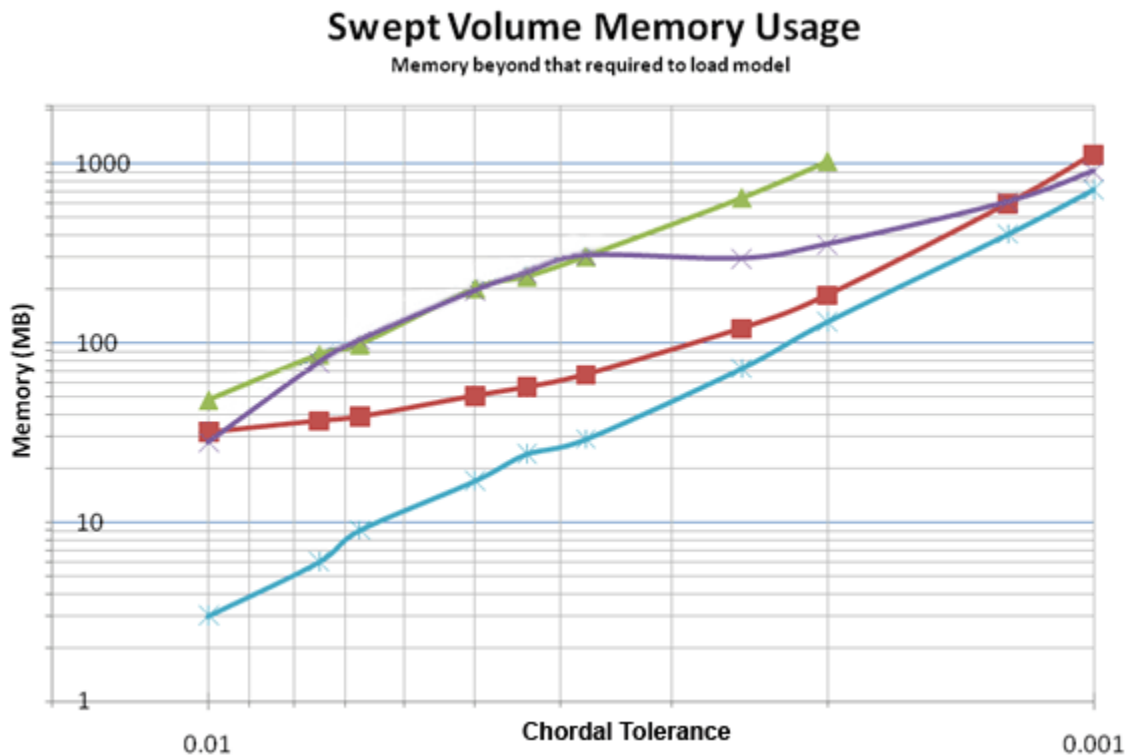
A large amount of physical memory is used when generating high-resolution swept volume envelopes over complex sweeping paths. 64-bit operating environments allow the application to make better use of available physical memory, which it requires in several large, contiguous chunks. This can be an issue on 32-bit systems, as their virtual address space often becomes highly fragmented, which causes the swept volume generation to fail. A system with at least 2 GB of RAM is recommended. Running a 64-bit operating system is also recommended.

For swept volume generation, the application runs most efficiently within the bounds of physical memory, and performance decreases dramatically when the system begins to use the swap space. Accordingly, the application adheres to the memory percentage specified in the **global memory budget**.

Chordal tolerance

Swept volume fidelity or visual quality is controlled with the chordal tolerance value, which is specified in the **Create Swept Volume** dialog box. The application produces a polygonal envelope positioned no further from the exact swept volume envelope than this tolerance value, which is always in the unit of measurement of the currently loaded model. Note that the generated polygonal envelope may be either inside or outside the exact swept volume envelope.

When generating swept volumes, the application's peak memory consumption is not closely related to the size of the input model or the complexity of the sweeping path. Memory usage depends most strongly on the specified chordal tolerance. The graph below shows the memory required to generate swept volumes, in addition to the memory needed to load the model, for a variety of models and quality settings. Generally, with the chordal tolerance set to medium quality, for most models the application consumes an additional 250 MB during the generation of swept volumes.



Generate swept volumes from moving parts

Use motion files to create swept volumes from moving parts.

1. Choose **Actions**→**Swept Volume**→**Create**.
2. In the **Create Swept Volume** dialog box, type a name for the swept volume.
3. Click **Dynamic**.
4. Click **Browse**, and then, in the **Open Motion File** dialog box, from the **Files of type** list, select the type of motion file that you want to open. Browse and select the motion file, and then click **Open**.
5. Adjust the visual quality of the generated swept volume in one of these ways:
 - Click and drag the **Tessellation Quality** slider. Dragging the slider left towards **Low** reduces the amount of triangles in the swept volume while moving it towards **High** increases them.
 - In the **Chordal Tolerance** box, type a value between *0.05* and *0.001*.
6. Click **Create**.

The **Swept Volume Generation** dialog box appears, displaying the progress of the swept volume as it is created.

The new swept volume is inserted into the active session as a new model in the assembly tree.

7. (Optional) Save the swept volume.

Edit the input path of swept volumes

1. **Open the motion file** containing the motions paths you want to edit.
2. Edit the motion file in any of the following ways:
 - **Change the part assignments**
 - **Add, edit, or delete paths**
 - **Add, edit, or delete key frames**
3. **Save** the motion file.
4. **Generate** the swept volume.

Generate swept volumes from stationary parts

1. Choose **Actions**→**Swept Volume**→**Create**.
2. In the **Create Swept Volume** dialog box, type a name for the swept volume.
3. Click **Static**.
4. Specify the parts to use for the swept volume in one of these ways:
 - From the **Input Parts** list, select **Visible**, and then display only the parts that you want the swept volume to encompass.
 - Select **Selected**, and then select the parts or assemblies that you want the swept volume to encompass.
5. Adjust the visual quality of the generated swept volume in one of these ways:
 - Click and drag the **Tessellation Quality** slider to adjust the visual quality of the generated swept volume. Dragging the slider left towards **Low** reduces the amount of triangles in the swept volume while moving it towards **High** increases them.
 - In the **Chordal Tolerance** box, type a value between 0.05 and 0.001.
6. Click **Create**.

The **Swept Volume Generation** dialog box appears, displaying the progress of the swept volume as it is created.

The new swept volume is inserted into the active session as a new model in the assembly tree.

7. (Optional) Save the swept volume.

Specify swept volume preferences

Control the appearance of swept volumes with the swept volume preferences.

1. Choose **Actions→Swept Volume→Preferences**.
2. In the **Swept Volume Preferences** dialog box, adjust any of the following settings:

To	Do this
Adjust the transparency.	Click and drag the Transparency slider to the left or right.
Choose the color.	Do one of the following: a. Click Use specified color. b. Click the color well and choose a color.
Use a random color.	Click Use arbitrary color .

3. Click **OK**.

Save swept volumes

Export swept volumes as 3D models. You also can create .vf session files or .plmxml files to preserve the state of the current work session.

Note:

You must export a swept volume as a 3D model before saving a session or .plmxml file for the swept volume to be properly referenced when the session or .plmxml file is reloaded.

Do any of the following:

To	Do this
Export swept volumes as 3D models	1. Display the swept volume. 2. Choose File→Export.

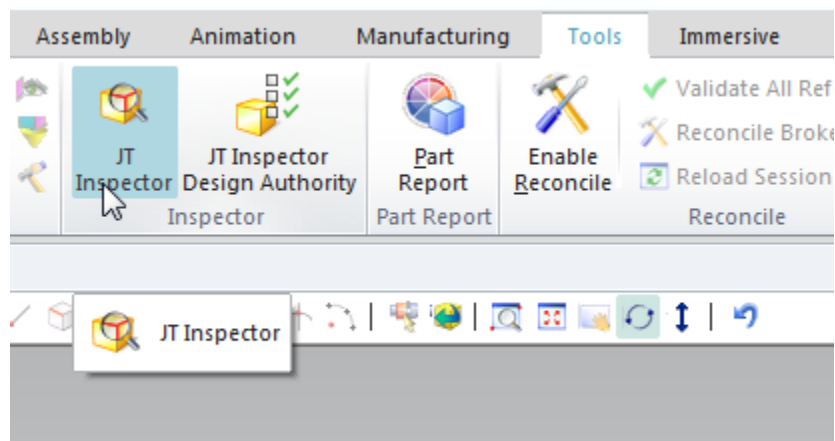
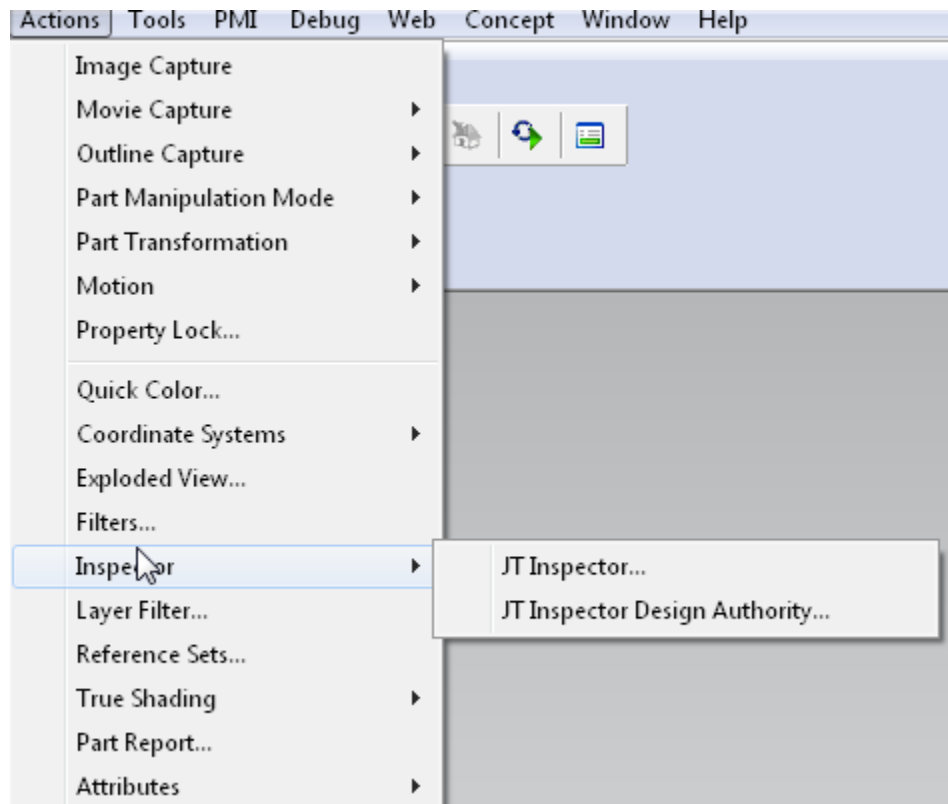
To	Do this
	3. Export the swept volume as a 3D model.
Create a session file	1. Display the swept volume. 2. Choose File→Export. 3. Export the swept volume as a 3D model. 4. Choose File, and then select a session save option.
Create a .plmxml file	1. Display the swept volume. 2. Choose File→Export. 3. Export the swept volume as a 3D model. 4. Choose File→Preferences→PLM XML. 5. In the PLM XML Preferences dialog box, on the Save page, select the Save Inserted Assemblies check box. 6. Click OK. 7. Choose File→Save As. 8. In the Save File dialog box, in the File name box, type a name for the file. 9. Click Save.

Inspecting JT files

What is JT Inspector?

JT Inspector examines and reports on the content of JT files, validates the integrity of its data structures, and checks for fidelity against the source CAD data. *JT Inspector Design Authority* extends the JT Inspector base capabilities by expanding the JT Fidelity to include *Cloud Of Points*, *PMI*, and *Model View Fidelity* tests. In addition, JT Inspector Design Authority introduces a new category of tests – SASIG Integrity, to check the precise geometry against the SASIG (Strategic Automotive Product Data Standards Group) standards.

Inspector module contains the JT Inspector(basic) and JT Inspector Design Authority and can be invoked using the menu below.



You can use Inspector to test and confirm that JT files are fit for purpose, particularly where JT is used in business critical processes.

- Where JT quality is critical to the quality of designed and manufactured products
- Where JT is approved for use within the organization
- Where JT files are exchanged in the manufacturing and design supply chain
- Where JT is stored as an archive of the published design

Inspector runs either from the command line to produce an XML report or from the user interface within Lifecycle Visualization to produce a result panel. You can include the Inspector command line executable as part of an automated process to review JT files. You can use Inspector within Lifecycle Visualization to examine more closely specific JT files from that process.

Help for the command line component is included in the utility. Type `<jtinspector -help>` to list the available controls. Sample configuration files `inspector.config`, `inspectorDesignAuthority_NAS9300.config`, and `inspectorDesignAuthority_MIL31000A.config` are located at `Installation_directory\etc\Inspector\`.

With Inspector, you specify which parts and assemblies to inspect, and which tests to run.

JT Inspector Design Authority offers four categories of tests. JT Content reports on what is in a JT file. JT Integrity and SASIG Integrity, focus on the downstream usability of JT content. JT Fidelity tests focus on how accurately the JT content matches the CAD system.

- **JT Content**
- **JT Integrity**
- **JT Fidelity**
- **SASIG integrity**

Test results within Lifecycle Visualization are always displayed on the monitor. If geometry failures are found, you can select the entity from the results, highlight the geometry, and zoom to the failure. You can also save the results to a file for future reference. You can save a formatted HTML report to serve as a quality record. Alternatively, you can also save an Encoded XML report that includes all the results – summary as well as details.

Licensing JT Inspector Design Authority

JT Inspector Design Authority requires an additional license (license token = "jt_inspect_workflow") to allow user to run the following additional tests.

- *JTFidelity*
 - Cloud Of Points
 - PMI
 - Model Views
- *SASIGIntegrity*
 - All tests under this category require JT Inspector Design Authority license.

JT Content

These tests inspect and report on B-Rep, tessellation, PMI, metadata properties, and physical properties. You can, for example, determine a part's B-Rep Entity ID, and report the number of bodies, faces, edges, and triangles stored for each tessellation Level of Detail. The tests report on the quantity of each PMI type and metadata.

JT Content section appears in the configuration file as follows:

```
JTContent {  
  preciseGeometry = true  
  tessellation = true  
  pmi = true  
  modelViews = true  
  metaProperties = true  
  physicalProperties = true  
  physicalPropertiesAccuracy = 0.99  
}
```

JT Integrity

JT Integrity tests report on the downstream usability of the persisted JT content.

Precise Geometry Integrity tests use Parasolid to validate any XT B-Rep geometry contained in the part. An ordered series of tests validates the topology and geometry of each geometric entity. The inspection report identifies any invalid geometry, missing geometry, curves, surfaces, edges, faces, and reports overall geometry/topology consistency.

PMI Integrity tests validate content of the PMI data. It also validates that geometric associations are present and that polyline data, polygon data, and persisted IDs all exist.

JT Integrity appears in the configuration file as follows:

```
JTIntegrity {  
  preciseGeometry = true  
  pmiGraphical = true  
  pmiAssociations = true  
}
```

JT Fidelity

JT Fidelity tests inspect JT files for how accurately the JT content matches the CAD content.

JT Fidelity appears in the configuration file as follows:

```
JTFidelity {
  numberOfComponents = true
  componentPositionOrientation = true

  surfaceArea = true
  surfaceAreaFidelityThreshold = 0.05

  mass = true
  massFidelityThreshold = 0.05

  volume = true
  volumeFidelityThreshold = 0.05

  density = true
  densityFidelityThreshold = 0.05

  centerOfGravity = true
  centerOfGravityThreshold = 0.01

  cloudOfPoints = true
  pointFidelityThreshold = "0.01 millimeters"

  pmiCounts = true

  modelViewCounts = true
  modelViewProperties = true

  modelViewPMIDisplay = true

  pmiGraphicalDisplay = true

  pmiPolylineLengthDeviation = 0.05

  pmiCentroidDeviation = 0.05

  associatedAttachedGeometry = true

  associatedAttachedGeomDeviation = 0.05

  modelViewCameraDeviation = 5
  modelViewPMIDisplayDeviation = 0.001

  pmiSemanticDefinition = true
}
```

Note:

The cloudOfPoints test, PMI tests, and Model View tests are only available if you invoke the JT Inspector Design Authority feature.

The mass property fidelity tests — surface area, volume, center of gravity, density, and mass — use Parasolid to calculate physical properties of the precise XT B-Rep geometry stored in the JT parts. These calculated properties are compared against corresponding CAD validation properties.

CAD validation properties required to be populated are JT_PROP_MEASUREMENT_UNITS, CAD_MASS_UNITS, CAD_SURFACE_AREA, CAD_VOLUME, CAD_CENTER_OF_GRAVITY, and CAD_DENSITY.

JT Property Name	Meaning	Data Type	Values
JT_PROP_MEASUREMENT_UNITS	CAD Model Units	String	millimeters centimeters meters inches feet yards micrometers decimeters kilometers mils miles
CAD_MASS_UNITS	Units of mass	String	
CAD_SURFACE_AREA	Surface area of solids and sheets within part	float	numeric
CAD_VOLUME	Volume of solids within part	float	numeric
CAD_DENSITY	Density of solids within part	float	numeric
CAD_MASS	Mass or weight of solids within part	float	numeric
CAD_CENTER_OF_GRAVITY	Center of gravity of solids within part	String	3 space separated numerics

Note:

CAD_SURFACE_AREA is in $\text{JT\_PROP\_MEASUREMENT\_UNITS}^2$

CAD_VOLUME is in $\text{JT\_PROP\_MEASUREMENT\_UNITS}^3$

CAD_MASS is in CAD_MASS_UNITS

CAD_DENSITY is in $\text{CAD\_MASS\_UNITS} / \text{JT\_PROP\_MEASUREMENT\_UNITS}^3$

CAD_CENTER_OF_GRAVITY is in JT_PROP_MEASUREMENT_UNITS

For CAD_DENSITY, it is assumed that all bodies in the part are of equal density.

JT Fidelity also includes the Cloud Of Points test. This involves point fidelity checking of precise XT B-Rep geometry against point cloud data. The JT Inspector reads point cloud data from a .plmxml file, from the validation data within a STEP or STL or .plmxml file, or infers points from the tessellation vertices of a second JT file. Fidelity is reported against a user-supplied tolerance to be specified in millimeters.

PMI and Model View Fidelity checking is also included in the JT Fidelity checking. The PMI Fidelity tests include the total PMI counts. The PMI counts per type in the JT file are compared with the CAD validation properties and any differences in PMI counts are reported.

The Model View Fidelity tests include the total Model View counts, Model View properties per Model View, and PMI Content per Model View. The Model Views in the JT file are compared with the CAD validation properties and any differences in the Model Views are reported.

The CAD validation properties required to be populated are:

On Assembly Node:

Property Name	Type/Format	Description	Prerequisites
CAD_CHILD_INSTANCE_COUNT	Int	Number of components in the assembly. Each node which is the parent node of at least one instance will have a property attached to enumerate the actual number of child instances of that node. This number will be defined by the sending system.	Set CAD_FIDELITY_VERSION = 3.1 or higher
CAD_NOTIONAL_SOLID_CENTROID	JtString formatted as three floating point	The component position and orientation.	Set CAD_FIDELITY_VERSION = 3.1 or higher

Property Name	Type/Format	Description	Prerequisites
	values separated by whitespace	For the top node and each intermediate node of a product structure, a notional solid is assumed within the child node of each child instance of that node. Using the positional and orientation relationship for each child instance, a centroid position can be calculated for the combined set of notional solids within the set of child instances. The notional solid will be a cube of size 10 x 10 x 10. The notional solid will be positioned with its centroid at (10.0, 10.0, 10.0) of the coordinate system of the child node. Note that the actual size and shape of the notional solid will not, in fact, affect the overall result. The key data is the centroid position and the assumption that the volume of the notional solid in each child node is the same. Mathematically, the calculated point is the mean of the set of points at (10.0, 10.0, 10.0) within the child nodes.	

On Part Node:

Property Name	Type/Format	Description	Prerequisites
PMI_FIDELITY_VERSION	string	Identifies the schema used for the PMI, Model Views and Geometry, fidelity properties. The current version is 3 The following are all the supported versions and its associated TcVis versions: PMI_FIDELITY_VERSION = 1 requires Teamcenter Visualization 11.3 or later. PMI_FIDELITY_VERSION = 2 requires Teamcenter Visualization Version 12.0 or later PMI_FIDELITY_VERSION = 3 requires Teamcenter Visualization Version 12.2 or later.	Requires Teamcenter Visualization 11.3 or later
CAD_FIDELITY_VERSION	string	Identifies the schema used for the PMI, Model Views, Assembly and Geometry fidelity properties. It is identified by a number that has a major and minor versions. The current version is 3.2. In the current version, the major version is 3 and the minor version is 2.	Requires Teamcenter Visualization 13.2 or later
CAD_PMI_COUNT	int	A total count of PMI entities indicated in the source CAD system.	Set PMI_FIDELITY_VERSION =1 or higher
CAD_PMI_COUNT_-BALLOON	int	A count of PMI entities of a particular type as indicated in the source	Set PMI_FIDELITY_VERSION =1 or higher

CAD_PMI_COUNT_- CENTERLINE	CAD system. The count for any PMI types not mentioned is assumed to be zero.
CAD_PMI_COUNT_- COORDINATENOTE	
CAD_PMI_COUNT_DAT UMFEATURESYMBOL	
CAD_PMI_COUNT_- DATUMTARGET	
CAD_PMI_COUNT_- DIMENSION	
CAD_PMI_COUNT_- CALLOUTDIMENSION	
CAD_PMI_COUNT_- CHAMFERDIMENSION	
CAD_PMI_COUNT_FEAT URECONTROLFRAME	
CAD_PMI_COUNT_- DATUMLOCATOR	
CAD_PMI_COUNT_- NOTE	
CAD_PMI_COUNT_USE RDEFINEDSYMBOL	
CAD_PMI_COUNT_MAT ERIALSPECIFICATION	
CAD_PMI_COUNT_PRO CESSSPECIFICATION	
CAD_PMI_COUNT_- PARTSPECIFICATION	
CAD_PMI_COUNT_- EMARKING	
CAD_PMI_COUNT_- REGION	
CAD_PMI_COUNT_- SURFACEFINISH	
CAD_PMI_COUNT_- LINEWELD	
CAD_PMI_COUNT_- SPOTWELD	

CAD_PMI_COUNT_EDGECONDITION	int	A count of PMI entities of a particular type as indicated in the source CAD system. The count for any PMI types not mentioned is assumed to be zero.	Set CAD_FIDELITY_VERSION=3.2 or higher
CAD_MV_COUNT	int	A count of modelviews indicated in the source CAD system.	Set PMI_FIDELITY_VERSION=1 or higher

On PMI Node:

Property Name	Type/Format	Description	Prerequisites
CAD_PMI_CENTROID	JtString formatted as three floating point values separated by whitespace	Mathematical centroid of the PMI presentation polylines indicated by the source CAD system. In the source CAD system, a centroid shall be calculated for PMI, based on the following algorithm: - Each Polyline annotation consists of line segments and circular arcs. - The "centroid" of a line segment is the center point of the line. - The "centroid" of a circular arc is on the radius joining the center of the arc at the position $\frac{\text{arc angle}}{\sin(\frac{\text{arc angle}}{2})} \cdot \frac{\text{radius}}{2}$. - For a set of line segments and arcs, the "centroid" is arrived at by	Set PMI_FIDELITY_VERSION=1 or higher

Property Name	Type/Format	Description	Prerequisites
		a composition of all the element centroids, with the length of each line segment or arc as weight.	
CAD_PMI_POLYLINE_LENGTH	JtString formatted as a single floating point value		Set PMI_FIDELITY_VERSION =1 or higher
CAD_PMI_ASSOCIATED_GEOM_AREA	JtString formatted as a single floating point value followed by a linear unit (e.g., 'm' instead of 'm ² ')	Amount of surface area of the part's BRep associated with the PMI in the source CAD system.	Set PMI_FIDELITY_VERSION =1 or higher Note: PMI – Body/ Component associated/ attached area was not included in the first release of CAD_PMI_ASSOCIATED_GEOM_AREA/ CAD_PMI_ATTACHED_GEOM_AREA attributes, PMI_FIDELITY_VERSION should be bumped up to '2' if this validation property includes the area of the associated/ attached bodies/ components.
CAD_PMI_ASSOCIATED_GEOM_LENGTH	JtString formatted as a single floating point value followed by a linear unit	Amount of curve length associated with the PMI in the source CAD system.	Set PMI_FIDELITY_VERSION =1 or higher
CAD_PMI_ATTACHED_GEOM_AREA	JtString formatted as a single floating point value followed by a linear unit	Amount of surface area of the part's BRep attached to the PMI in the source CAD system.	Set PMI_FIDELITY_VERSION =1 or higher

Property Name	Type/Format	Description	Prerequisites
			Note: PMI – Body/ Component associated/ attached area was not included in the first release of CAD_PMI_ASSOCIATED_ GEOM_AREA / CAD_PMI_ATTACHED_ GEOM_ AREA attributes, PMI_FIDELITY_VERSION should be bumped up to '2' if this validation property includes the area of the associated/ attached bodies/ components.
CAD_PMI_ATTACHED_GEOLENGTH	JtString formatted as a single floating point value followed by a linear unit	Amount of curve length attached to the PMI in the source CAD system.	Set PMI_FIDELITY_VERSION =1 or higher
CAD_PMI_STRING	JtString representing the semantic definition of the part	String syntax is the same as ISO-10303 Unicode String specification - http://cax-if.de/documents/rec_%C2%ADprac_unicode_strings_rev%C2%AD_j_2011-05-25.pdf	Set PMI_FIDELITY_VERSION =1 or higher
CAD_PMI_ASSOCIATED_PMI_ORIGINS Type: JtString formatted as semi-colon delimited list of origin values followed by a linear unit. For example: CAD_PMI_ASSOCIATED	Type: JtString formatted as semi-colon delimited list of origin values followed by a linear unit. For example: CAD_PMI_ASSOCIATED_PMI_ORIGINS = (0,0,0); (10,0,0) mm	For every PMI, JT Inspector will retrieve all of its associated PMI and if the origins of these PMI match completely with the origin values listed in the validation property,	Set PMI_FIDELITY_VERSION =2 or higher

Property Name	Type/Format	Description	Prerequisites
_PMI_ORIGINS = (0,0,0); (10,0,0) mm		then the PMI will pass this test.	
CAD_PMI_ATTACHED_PMI_ORIGINS Type: JtString formatted as semi-colon delimited list of origin values. For example: CAD_PMI_ASSOCIATED_PMI_ORIGINS = (0,0,0); (10,0,0) mm	Semi-colon delimited list of destination PMI origin values that the source PMI is attached to.	For every PMI, JT Inspector will retrieve all of its attached PMI and if the origins of these PMI match completely with the origin values listed in the validation property, then the PMI will pass this test.	Set PMI_FIDELITY_VERSION =2 or higher

On Model Views:

Property Name	Type/Format	Description	Prerequisites
CAD_PMI_COUNT_-BALLOON CAD_PMI_COUNT_-CENTERLINE CAD_PMI_COUNT_-COORDINATENOTE CAD_PMI_COUNT_DATUMFEATURESYMBOL CAD_PMI_COUNT_-DATUMTARGET CAD_PMI_COUNT_-DIMENSION CAD_PMI_COUNT_-CALLOUTDIMENSION CAD_PMI_COUNT_-CHAMFERDIMENSION CAD_PMI_COUNT_FEATURECONTROLFRAME CAD_PMI_COUNT_-DATUMLOCATOR CAD_PMI_COUNT_-NOTE	int	A count of PMI entities of a particular type as indicated in the source CAD system. The count for any PMI types not mentioned is assumed to be zero.	Set PMI_FIDELITY_VERSION =2 or higher

CAD_PMI_COUNT_- USERDEFINEDSYMBOL			
CAD_PMI_COUNT_MAT ERIALSPECIFICATION			
CAD_PMI_COUNT_PRO CESSSPECIFICATION			
CAD_PMI_COUNT_- PARTSPECIFICATION			
CAD_PMI_COUNT_- EMARKING			
CAD_PMI_COUNT_- REGION			
CAD_PMI_COUNT_- SURFACEFINISH			
CAD_PMI_COUNT_- LINEWELD			
CAD_PMI_COUNT_- SPOTWELD			
CAD_PMI_COUNT_EDG ECONDITION	int	A count of PMI entities of a particular type as indicated in the source CAD system. The count for any PMI types not mentioned is assumed to be zero.	Set CAD_FIDELITY_VERSION=3.2 or higher
CAD_MV_STRING	JtString representing the semantic definition of the model view	It includes the Model View identifier, type, name and color. For example, VIEW\wVM\w"Front" \wfffff f0b478 2fa866 000000	Set PMI_FIDELITY_VERSION=2 or higher
CAD_MV_CAMERA_- POINTS	JtString representing the camera for a model view.	It includes the list of pairs of points which should be comma separated and the pairs should be semi-colon separated. For example, 0 0 0, 0 0 0; 10 0 0, 0.25 0 0; 10 10 0. 0.25 0.25 0; 0 10 0, 0 0.25	Set PMI_FIDELITY_VERSION=2 or higher

		0; 0 0 10, 0 0 0.25; 10 0 10, 0.25 0 0.25; 10 10 10, 0.25 0.25 0.25; 0 10 10, 0 0.25 0.25;	
CAD_MV_ASSOCIATED_PMI_ORIGINS	Semi-colon delimited list of PMI origin values.	The origins are stored in the property in the following format: (x1, y1, z1); (x2, y2, z2);...	Set PMI_FIDELITY_VERSION =3 or higher

SASIG Integrity

Strategic Automotive Product Data Standards Group (SASIG) has a recommended set of guidelines for assessing Product Data Quality (PDQ). SASIG Integrity section allows for a series of precise geometry checks to be performed on the XT B-Rep. A sub-set of these same checks are recommended by the NAS 9300-110 and MIL-STD-31000A standards.

Note:

The SASIGIntegrity tests are only available if you invoke the JT Inspector Design Authority feature.

SASIG Integrity appears in the configuration file as follows:

```
SASIGIntegrity {
  largeCurveGap(G-CU-LG) = true
  distanceTol(G-CU-LG) = "0.001 millimeters"

  nonTangentCurves(G-CU-NT) = true
  angleTol(G-CU-NT) = "3.0 degrees"

  nonSmoothSurfacesOrPatches(G-SU-NS) = true
  curvatureTol(G-SU-NS) = "0.1"

  narrowSurfaceOrPatch(G-SU-NA) = true
  distanceTol(G-SU-NA) = "0.005 millimeters"

  degenerateSurfaceBoundary(G-SU-DC) = true
  distanceTol(G-SU-DC) = "0.01 millimeters"

  indistinctSurfaceKnots(G-SU-IK) = true
  knotTol(G-SU-IK) = "0.000001"

  selfIntersectingSurface(G-SU-IS) = true

  excessivelyHighDegreeSurface(G-SU-HD) = true
  maximumDegree(G-SU-HD) = "5"
```



```
smallSurfaceRadiusOfCurvature(G-SU-CR) = true
minimumCurvatureRadius(G-SU-CR) = "0.05 millimeters"
```

```
relativelyNarrowNeighboringPatches(G-SU-RN) = true
relativeTolerance(G-SU-RN) = "0.01"
```

```
degenerateSurfaceCorner(G-SU-DP) = true
cornerAngleTolerance(G-SU-DP) = "10.0 degrees"
```

```
embeddedSurfaces(G-SU-EM) = true
maxSurfaceCount(G-SU-EM) = "2"
```

```
unusedPatches(G-SU-UN) = true
```

```
foldedSurface(G-SU-F0) = true
flappingAngleTolerance(G-SU-F0) = "90.0 degrees"
```

```
wavySurface(G-SU-WV) = true
maxCurvatureSignChanges(G-SU-WV) = "3"
```

```
intersectingShells(G-S0-IS) = true
```

```
multiVolumeSolid(G-S0-MU) = true
```

```
solidVoid(G-S0-V0) = true
```

```
tinySolid(G-S0-TI) = true
volTolerance(G-S0-TI) = "0.01 millimeters"
```

```
largeEdgeFaceGap(G-FA-EG) = true
faceGapTol(G-FA-EG) = "1.0 millimeters"
```

```
tinyFace(G-FA-TI) = true
maxAreaSize(G-FA-TI) = "0.00001 millimeters"
```

```
largeVertexGap(G-FA-VG) = true
vertexGapTol(G-FA-VG) = "1.0 millimeters"
```

```
embeddedFace(G-FA-EM) = true
embeddedFaceTol(G-FA-EM) = "0.01 millimeters"
```

```
intersectingLoop(G-FA-IS) = true
```

```
inconsistentFaceOnSurface(G-FA-IT) = true
```

```
multiRegionFace(G-FA-MU) = true;
```

```
narrowFace(G-FA-NA) = true
narrowFaceTol(G-FA-NA) = "0.01 millimeters"
```

```

narrowRegion(G-FA-RN) = true
narrowRegionTol(G-FA-RN) = "0.01 millimeters"

largeEdgeGap(G-LO-LG) = true
edgeGapTol(G-LO-LG) = "1.0 millimeters"

nonTangentEdges(G-LO-NT) = true
angleTol(G-LO-NT) = "3.0 degrees"

nonSmoothEdges(G-LO-NS) = true
curvatureTol(G-LO-NS) = "0.1"

inconsistentEdgeInLoop(G-LO-IT) = true

sharpEdgeAngle(G-LO-SA) = true
angleTol(G-LO-SA) = "0.5 degrees"

selfIntersectingLoop(G-LO-IS) = true

freeEdge(G-SH-FR) = true

selfIntersectingShell(G-SH-IS) = true

inconsistentFaceInShell(G-SH-IT) = true

sharpFaceAngle(G-SH-SA) = true
angleTol(G-SH-SA) = "5.0 degrees"

largeFaceGap(G-SH-LG) = true
faceGapTol(G-SH-LG) = "1.0 millimeters"

overUsedEdge(G-SH-NM) = true
maxEdgeCount(G-SH-NM) = "2"

overUsedVertex(G-SH-OU) = true
maxEdgeCount(G-SH-OU) = "3"

embeddedSolids(G-SO-EM) = true
maxSolidCount(G-SO-EM) = "2"

fragmentedSurface(G-SU-FG) = true
maxPatchCount(G-SU-FG) = "2500"

tinySurfacePatch(G-SU-TI) = true
surfacePatchTol(G-SU-TI) = "0.00001 millimeters"

nonTangentSurface(G-SU-NT) = true
angleTol(G-SU-NT) = "3.0 degrees"

```

```

indistinctCurveKnots(G-CU-IK) = true
knotTolerance(G-CU-IK) = "0.00001"

smallRadiusOfCurvature(G-CU-CR) = true
minimumCurvatureRadius(G-CU-CR) = "0.05 millimeters"

nonSmoothCurves(G-CU-NS) = true
curvatureTol(G-CU-NS) = "0.1"

excessivelyHighDegreeCurve(G-CU-HD) = true
maxCurveDegree(G-CU-HD) = "5"

tinyEdge(G-ED-TI) = true
distanceTol(G-ED-TI) = "0.01 millimeters"

tinyCurveOrSegment(G-CU-TI) = true
curveLengthTol(G-CU-TI) = "0.01 millimeters"

selfIntersectingCurve(G-CU-IS) = true

fragmentedCurve(G-CU-FG) = true
maxSegmentCount(G-CU-FG) = "10"

wavyPlanarCurve(G-CU-WV) = true
maxWaveCount(G-CU-WV) = "3"

embeddedCurves(G-CU-EM) = true
identicalCurveDistanceTol(G-CU-EM) = "0.01 millimeters"

nonTangentFaces(G-SH-NT) = true
angleTol(G-SH-NT) = "0.5 degrees"

nonSmoothFaces(G-SH-NS) = true
curvatureTol(G-SH-NS) = "0.1"

fragmentedEdge(G-ED-FG) = true
maxSegmentCount(G-ED-FG) = "10"
inconsistentEdgeOnCurve(G-ED-IT) = true
}

```

Inspecting JT data

Introduction to configuring and inspecting JT data

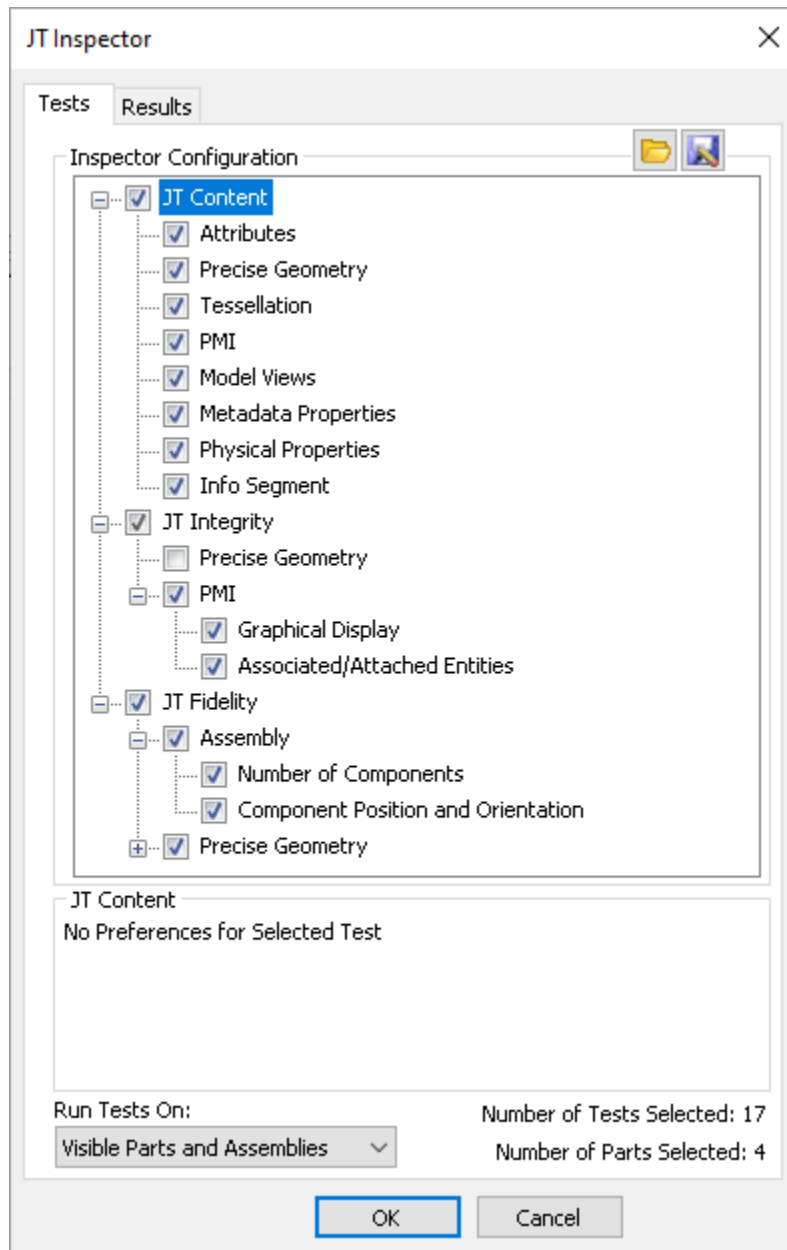
The **Inspector** dialog box has two tabs, **Tests** and **Results**. On the **Tests** tab, you configure and run the inspection tests.

Note:

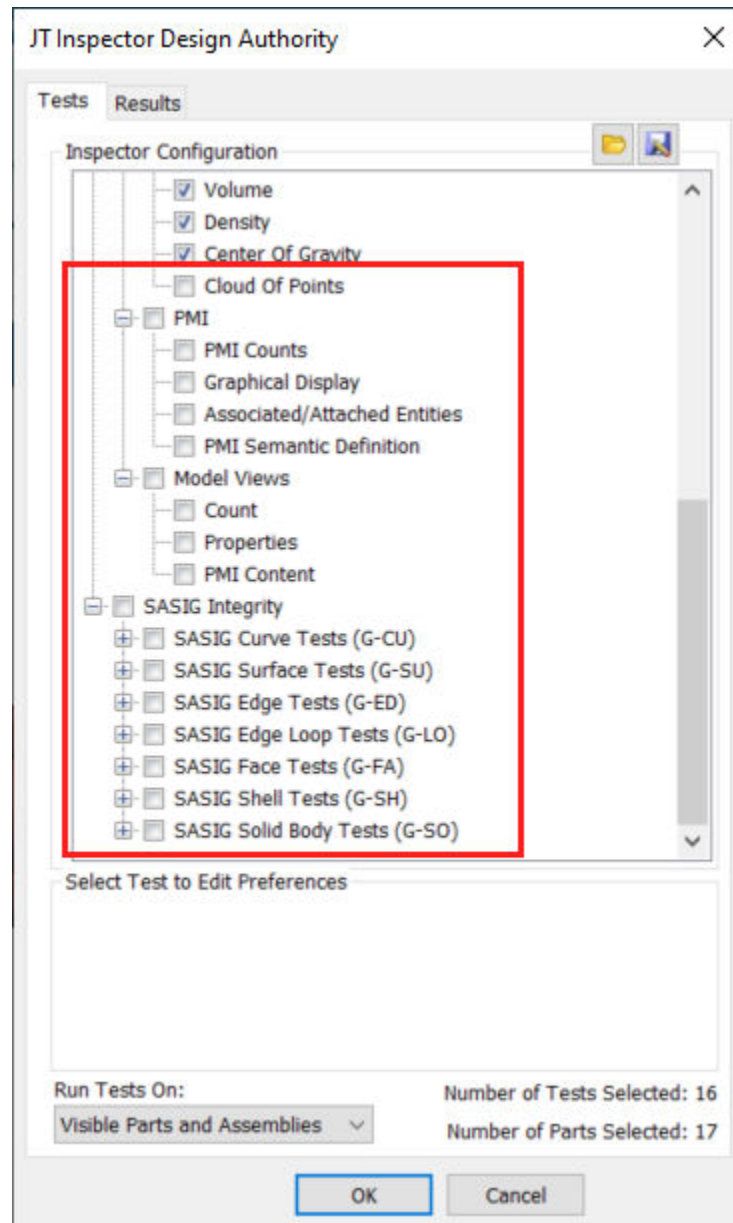
If you open several JT files with corresponding **Inspector** dialog boxes, the dialog box of the active file is displayed.

On the **Tests** tab, you can do the following:

- Load a configuration file. Sample configuration files are at <install_location>/etc/Inspector.
- In the **Inspector Configuration** section, **select the tests to run** and set test **preferences**.
- In the **Run Tests On** section, **specify the parts or assemblies to inspect**.
- Select **OK** to **run the inspection tests**.



If you invoke the JT Inspector Design Authority feature, additional tests are available.



Set inspection preferences using configuration files

You can use configuration files to set inspection preferences and specify which inspection tests to run. Configuration files simplify running the same tests on multiple JT files.

A sample configuration file, **inspector.config**, is located here: `<Installation_directory>\etc\Inspector\`.

1. (Optional) Create a configuration file using a text editor.
2. Choose **Actions**→**Inspector**→**JT Inspector**.

3. In the **Inspector** dialog box, on the **Tests** tab, click  to load a configuration file.
4. Click **Load Test Configuration**.
5. Choose a configuration file and click **Open**.

The **Tests** tab updates with the tests and preferences specified in the configuration file.

6. To save the final settings to a configuration file, click .

Select the inspection tests to run

There are four categories of tests:

- **JT content**
- **JT Integrity**
- **JT Fidelity**
- **SASIG Integrity**

Some categories have multiple tests.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. In the **Inspector** dialog box, on the **Tests** tab, select or clear the check box in the **Categories** section to select the inspection tests to run.

Note:

Use the check boxes to determine the status of your choices.

- A check box with a standard green check-mark means the category or test is selected.
- A cleared check box means the category or test is not selected.
- A check box containing a gray check-mark means not all tests within that category are selected.

Set inspection test accuracy and error threshold

You can specify the **Calculation Accuracy** percentage for **Physical properties** tests, and **Error Threshold** percentage for all **Fidelity** tests except for **Center of gravity**.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. In the **Inspector** dialog box, on the **Tests** tab, do the following:

Highlight this test	Then do this
Physical properties	a. Highlighting the test displays Preferences at the bottom of the dialog box. b. Enter a percentage in Calculation Accuracy. Note: • Enter the value without the percentage sign. For instance, type 10 for 10%. • Percentage settings for physical properties indicate how accurate the calculation will be. Tests with higher percentage values take longer to process. • The physical properties lists the computed values using Parasolid library and only applies to XT B-Rep data. The accuracy setting helps interpret these computed values, and is not related to tessellation or LODs.
Surface Area Mass Volume Density Center Of Gravity	a. Highlighting the test displays Preferences at the bottom of the dialog box. b. Enter a percentage in Error Threshold. Note: A percentage setting for fidelity tests indicate how closely comparison values must match before the test considers the comparison a failure. Fidelity tests calculate the physical properties of XT B-Rep parts and compare them against the corresponding CAD physical properties.

Highlight this test	Then do this
Cloud of Points	a. Highlighting the test displays Preferences at the bottom of the dialog. b. In the Tolerances box, set a tolerance value, which is the distance in millimeters allowed from a perfect point in the point cloud. c. Browse to navigate to a point cloud file.
PMI	a. Highlighting the test displays Preferences at the bottom of the dialog. b. You may enter an external source of PMI fidelity Acceptable file types are <i>.plmxml</i> files.
GraphicalDisplay	a. Highlighting the test displays preferences at the bottom of the dialog. b. You may enter PMI centroid deviation. c. You may enter PMI polyline length deviation.
Associated/Attached Entities	a. Highlighting the test displays Preferences at the bottom of the dialog. b. You may enter Associated/attached geometry size deviation.
Properties	a. Highlighting the test displays preferences at the bottom of the dialog. b. You may enter Model View Camera Deviation.
PMI Content	a. Highlighting the test displays preferences at the bottom of the dialog. b. You may enter Model View PMI Display Deviation.
Any SASIG Integrity Test	a. Highlighting the test displays Preferences at the bottom of the dialog. b. Enter a Tolerance for the test. Linear tolerances are specified in millimeters and Angular tolerances are to be specified in degrees.

Specify parts and assemblies to inspect

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. In the **Inspector** dialog box, on the **Tests** tab, specify the parts or assemblies to inspect. From the **Run Tests On:** list, select one of the following:

To	Select
Inspect all visible parts and assemblies.	Visible Parts and Assemblies This is the default setting.
Inspect all selected parts and assemblies.	Selected Parts and Assemblies When you select an assembly, you include subassemblies and parts.
Inspect all assembly, subassembly, and parts in the file.	All Parts and Assemblies

Run inspection tests

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. In the **Inspector** dialog box, on the **Tests** tab, specify the parts to inspect.
3. Do one of the following to specify which tests to run:
 - Select or clear the check boxes to select the tests and (optionally) set JT Inspector preferences.
 - Load a test configuration file.
4. Click **OK**.

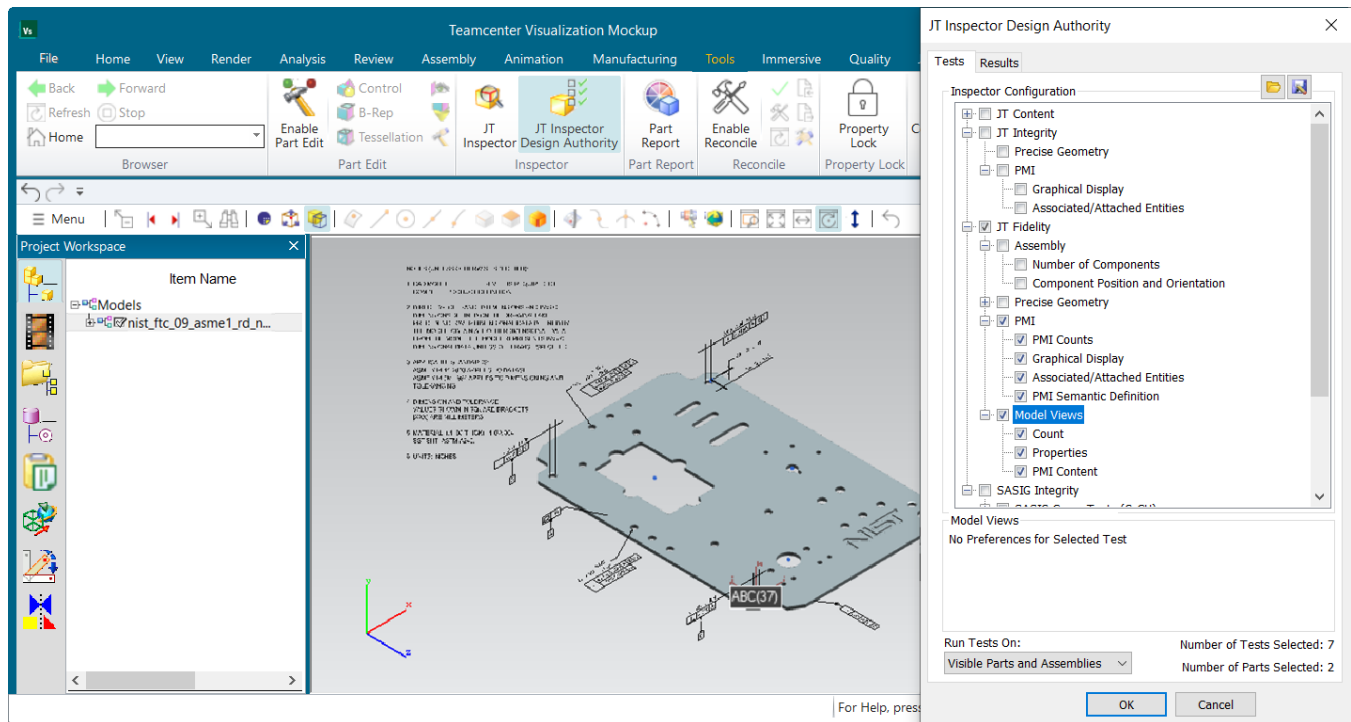
When the test is finished, the **Results** tab appears.

Configuring PMI checking in JT Inspector

JT Fidelity checking includes R and Model View fidelity. The 'Tests' tab of JT Inspector can be used to enable these tests. 'Run Tests On' drop-down allows for enabling the test on Visible Parts and Assemblies, Selected Parts and Assemblies or All Parts and Assemblies.

PMI/Model View Test Description

Category	Test	Description
PMI	PMI Counts	When enabled, JT Inspector checks the PMI counts in the JT files against the PMI content in the source CAD system. JT Inspector checks total PMI Counts as well as count of each individual PMI type.
	Graphical Display	When enabled, JT Inspector checks the PMI graphical display in the JT file against the graphical display in the source CAD system. Centroid and Polyline curve lengths are used to compare the graphical display between the JT and CAD system.
	Associated/Attached Entities	When enabled, JT Inspector checks that face surface area or the edge length of the geometry that the PMI is associated to and attached to is the same between the JT file and the source CAD system
	PMI Semantic Definition	When set to true, checks that the PMI semantic definition in the JT file is the same as the Semantic PMI definition in the source CAD system.
Model Views	Count	When set to enabled, JT Inspector checks the Model View counts in the JT files against the Model View counts in the source CAD system
	Properties	When enabled, JT Inspector checks that for each JT model view, the Model View identifier, Type, Name, Background Color and View Transformation is the same as that of the Model View in the source CAD system
	PMI Content	When enabled, JT Inspector checks that for each model view, the PMI counts in the Model View and the position of the PMI In the Model View matches that of the source CAD system.



Performing batch inspection of JT data

Inspect JT data from the command prompt

To run JT Inspector from command prompt, type **JTInspector <path_to_JT_file> -config <path_to_the_config_file> -output <path_to_output_xml_file>**.

Note:

- If you do not include a reference to your specific configuration file, the default configurations will be used.
- If you do not include a reference to an output XML file, the XML contents will be displayed on the console.
- In addition to the command line options documented below, all individual options documented in the configuration file can also be used as a command line options. This excludes the categories such as JTIntegrity, JTFidelity, SASIGIntegrity, and JTContent.

Enter command-line options, as required.

Use this option	To do this
-help	Display command line help
-output <pathname>	Set the file name and path of the output XML file
-config <configFile>	Use the specified configuration file instead of the installed configuration file
-preciseGeomJTContent	Include information about XT B-Rep(s) in the JT file
-preciseGeomIntegrity	Report on the validity of the Parasolid XT B-Rep and locate any faults.
-cloudOfPoints	Compare the translated geometry against an external file containing cloud of points generated from the source CAD system.
Note: This capability requires JT Inspector Design Authority license	
-pointFidelityThreshold <tolerance_with_units>	Specify a tolerance to use for the cloudOfPoints check
-pointOutput <path_to_JT_file>	Filename for the JT file of points that exceed tolerance. This file is generated only if necessary
-plmxmlPointOutput	Outputs a PLMXML file containing the cloud of points

Examples:

```
JTInspector.exe "C:\Temp\brique.jt" -config "C:\Temp\inspector.config" -output
JTInspectorOutput.xml -output JTInspectorOutput.html -preciseGeomIntegrity
```

```
JTInspector.exe "C:\Temp\brique.jt" -config "C:\Temp\inspector.config" -cloudOfPoints -compareFile
"C:\Temp\brique.stp" -pointFidelityThreshold 0.001mm -pointOutput PointCloudOutput.jt -output
JTInspectorOutput.xml
```

Sample Configuration file

```
version "EAITranslator" 1.0.0 "EAITranslator"

# JT Inspector Configuration
# This configuration can be copied and modified to control the tests
# conducted by
# JT Inspector at the command line.
# Additional tests conducted by JT Inspector Design Authority are not
# included here.
#
```

```

# Specify what JT Content to report
#JTContent {
  preciseGeometry = true
  tessellation = true
  pmi = true
  modelViews = true
  metaProperties = true
  physicalProperties = true
  physicalPropertiesAccuracy = 0.99
}

# Specify what JT Integrity to verify
#JTIntegrity {
  preciseGeometry = true
  pmiGraphical = true
  pmiAssociations = true
}

# Specify what JT Fidelity to compare
#
JTFidelity {
  surfaceArea = true
  surfaceAreaFidelityThreshold = 0.05

  mass = true
  massFidelityThreshold = 0.05

  volume = true
  volumeFidelityThreshold = 0.05

  density = true
  densityFidelityThreshold = 0.05

  centerOfGravity = true
  centerOfGravityThreshold = 0.01
}

#
# Specify what SASIG Integrity tests to perform
# Default settings conform to EN/NAS9300-110
SASIGIntegrity {

  largeCurveGap(G-CU-LG) = true
  distanceTol(G-CU-LG) = "0.001 millimeters"

  nonTangentCurves(G-CU-NT) = true
  angleTol(G-CU-NT) = "3.0 degrees"
}

```

```

largeEdgeFaceGap(G-FA-EG) = true
faceGapTol(G-FA-EG) = "1.0 millimeters"

tinyFace(G-FA-TI) = true
maxAreaSize(G-FA-TI) = "0.00001 millimeters"

largeVertexGap(G-FA-VG) = true
vertexGapTol(G-FA-VG) = "1.0 millimeters"

largeEdgeGap(G-LO_LG) = true
edgeGapTol(G-LO-LG) = "1.0 millimeters"

freeEdge(G-SH-FR) = true

largeFaceGap(G-SH-LG) = true
faceGapTol(G-SH-LG) = "1.0 millimeters"

overUsedEdge(G-SH-NM) = true
maxEdgeCount(G-SH-NM) = "2"

embeddedSolids(G-SO-EM) = true
maxSolidCount(G-SO-EM) = "2"

fragmentedSurface(G-SU-FG) = true
maxPatchCount(G-SU-FG) = "2500"

tinySurfacePatch(G-SU-TI) = true
surfacePatchTol(G-SU-TI) = "0.00001 millimeters"
}

```

Editing Configuration settings

Edit JTContent options

You can edit the options listed in this section to change how you inspect your JT files.

1. Open an existing configuration file with a text editor.
2. Edit the configuration file options.

Option	Keyword or value	Description
JTContent	JTContent {	JTContent {
preciseGeometry	true false	When set to <i>true</i> , include information about XT B-Rep(s) in the JT file. This also reports whether STT (Smart Topology Table) information is present in the JT file.

Option	Keyword or value	Description
tessellation	true false	When set to <i>true</i> , include information about the parameters used to tessellate the geometry in the file.
pmi	true false	When set to <i>true</i> include information about the PMI content in the JT file.
modelViews	true false	When set to <i>true</i> include information about the Modelviews content in the JT file.
physicalProperties	true false	When set to <i>true</i> , include information about physical properties (Surface Area, Volume, Mass, Density, Center of Gravity) in the JT file.
physicalPropertiesAccuracy	<percentage value> Ex: 0.99	If the calculation requested only involves simple geometries, an accurate method of integration is used, and the parameter " physicalPropertiesAccuracy " is ignored. In complex cases physicalPropertiesAccuracy is used to determine the accuracy of the calculation: it must be in the range 0.0 to 1.0, and as it approaches 1.0, the calculation will be more accurate but slower. The dependence of results on " physicalPropertiesAccuracy " is not linear, and values below 0.9 will give only coarse approximations. A great deal of calculation may be required to satisfy a physicalPropertiesAccuracy of 1.0, and in general convergence cannot be guaranteed. Supplying an accuracy parameter of 1.0 for parts involving B-surfaces is not recommended.
Close brace	}	}

Edit JTIntegrity options

Option	Keyword or value	Description
JTIntegrity	JTIntegrity {	JTIntegrity {
preciseGeometry	true false	When set to "true", check the integrity of the XT B-Rep(s) in the JT file. With JT version 10.5 or later, STT(Smart Topology Table) information is also checked. The XT B-Rep checker conducts a series of tests on the geometry and topology in the XT B-Rep. Details about the errors as well as the identifiers of the problematic bodies, faces, edges, curves and surfaces are written to the XML output file. When JT Inspector is run from within TC Visualization, the ability to "Fly to Selected" is also available.

Option	Keyword or value	Description
pmiGraphical	true false	When set to <i>true</i> , check the integrity of the PMI graphical data structures. PMI that fails this check will not be able to be displayed correctly.
pmiAssociations	true false	When set to <i>true</i> , checks that the PMI Associations can be resolved.
Close brace	}	}

Edit JTFidelity options

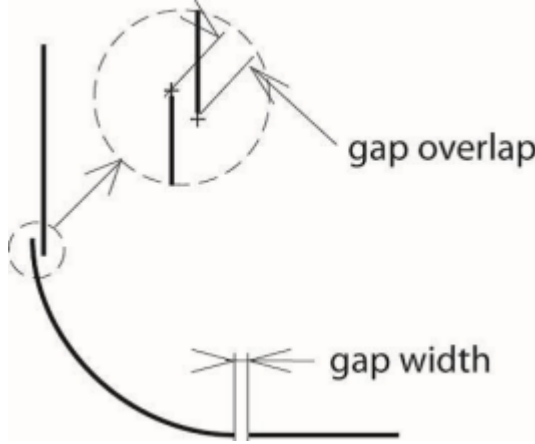
Option	Keyword or value	Description
JTFidelity	JTFidelity {	JTFidelity {
surfaceArea	true false	When set to <i>true</i> , check Surface Area fidelity. This means compare the surface area computed from the geometry against the value read from the CAD file.
surfaceAreaFidelityThreshold	<percent threshold value> For example: 0.05	Percentage deviation allowed before the fidelity check is marked as having failed. For example; when set to <i>0.05</i> , the fidelity check is marked as having failed if the computed value differs from the CAD value by more than 5% ($0.05 = 5\%$).
mass	true false	When set to <i>true</i> , check mass fidelity. This means compare the mass value computed from the geometry against the value read from the CAD file.
massFidelityThreshold	<percent threshold value> For example: 0.05	Percentage deviation allowed before the fidelity check is marked as having failed. For example; when set to <i>0.05</i> , the fidelity check is marked as having failed if the computed value differs from the CAD value by more than 5% ($0.05 = 5\%$)
volume	true false	When set to <i>true</i> , check volume fidelity. This means compare the volume computed from the geometry against the value read from the CAD file.
volumeFidelityThreshold	<percent threshold value> For example: 0.05	<i>0.05</i> , the fidelity check is marked as having failed if the computed value differs from the CAD value by more than 5% ($0.05 = 5\%$).

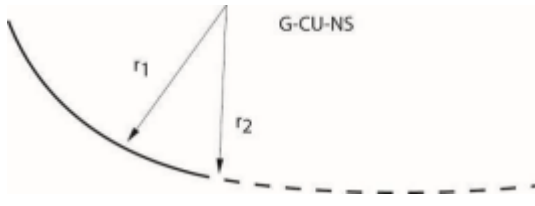
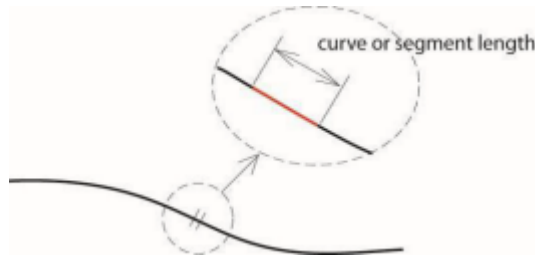
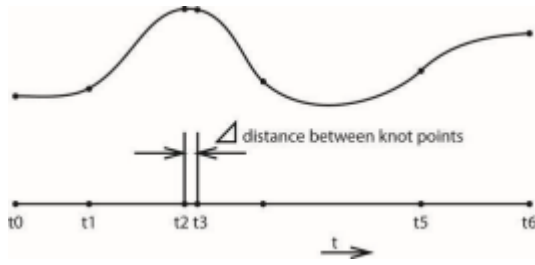
Option	Keyword or value	Description
density	true false	Check Density fidelity. This means compare the Density computed from the geometry against the value read from the CAD file.
densityFidelityThreshold	<percent threshold value> For example: 0.05	Percentage deviation allowed before the fidelity check is marked as having failed. For example; when set to 0.05, the fidelity check is marked as having failed if the computed value differs from the CAD value by more than 5% (0.05 = 5%)
centerOfGravity	true false	Check Center of Gravity fidelity. This means compare the center of gravity computed from the geometry against the value read from the CAD file.
centerOfGravityFidelityThreshold	<percent threshold value> For example: 0.05	Percentage deviation allowed before the fidelity check is marked as having failed. For example; when set to 0.05, the fidelity check is marked as having failed if the computed value differs from the CAD value by more than 5% (0.05 = 5%).
cloudOfPoints	true false	When set to <i>true</i> , compare the geometry against cloud of points generated from the CAD file.
pointFidelityThreshold	deviation threshold For example: 0.01 millimeters	Deviation allowed between the XT B-Rep geometry and a point from the point Cloud. For example, when the value is set to 0.01 millimeters, if the geometry deviates from the CAD point data by more than 0.01 millimeters, geometry has failed the fidelity test.
pmiCounts	true false	When set to <i>true</i> , checks the PMI counts in the JT files against the PMI content in the source CAD system.
pmiGraphicalDisplay	true false	When set to <i>true</i> checks the PMI graphical display in the JT file against the graphical display in the source CAD system. Centroid and Polyline curve lengths are used to compare the graphical display between the JT and CAD system.
pmiPolylineLengthDeviation	Allowed deviation value: For example: 0.05	Percentage deviation allowed before the test is considered failed. For example, when set to 0.05, if the total length of the polylines in the JT file for a given PMI deviates from the CAD polylines length for the PMI by more than 5% then the test is considered as failed.
pmiCentroidDeviation	Allowed deviation value:	Percentage deviation allowed before the test is considered failed. For example, when set to 0.05, if the centroid of the polylines in the JT file for a

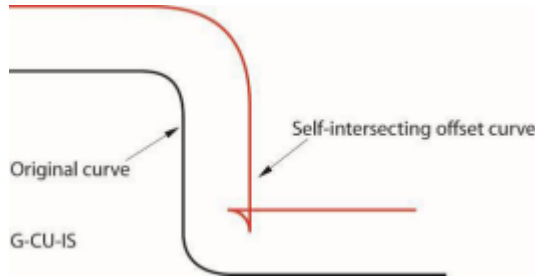
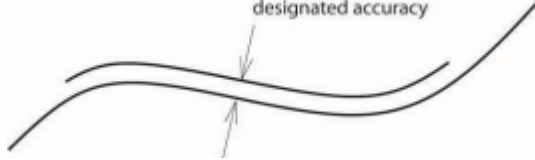
Option	Keyword or value	Description
	For example: 0.05	given PMI deviates from the CAD centroid for the PMI by more than 5% then the test is considered as failed.
associatedAttachedGeometry	true false	When set to <i>true</i> , checks that face surface area or the edge length of the geometry that the PMI is associated to and attached to is the same between the JT file and the source CAD system.
associatedAttachedGeom-Deviation	Allowed deviation value For example: 0.05	Percentage deviation allowed before the test is considered failed. For example, when set to 0.05, if the face surface area of the geometry associated with the PMI deviates by more than 5%, then the test is considered as failed.
pmiSemanticDefinition	true false	When set to true, checks that the PMI semantic definition in the JT file is the same as the Semantic PMI definition in the source CAD system.
modelViewCounts	true false	When set to <i>true</i> , checks the Model View counts in the JT files against the PMI content in the source CAD system.
modelViewProperties	true false	When set to true, checks per each model view, the Model View identifier, type, name, background color and view transformation against CAD system per Model View semantic definition.
modelViewPMIDisplay	true false	When set to true, checks per each model view, the number of PMI and origin of each PMI against the CAD system PMI counts and origin.
modelViewCameraDeviation	Allowed deviation value For example: 0.05	Percentage deviation allowed before the test is considered failed. For example, when set to 0.05, if the camera points deviate by more than 5%, then the test is considered as failed.
modelViewPMIDisplayDeviation	Allowed deviation value For example: 0.01	Percentage deviation allowed before the test is considered failed. For example, when set to 0.01, if the PMI origin deviate by more than 1%, then the test is considered as failed.
numberOfComponents	true false	When set to <i>true</i> , checks that the number of child instances for each assembly node in a JT file matches with the corresponding number in the source CAD file.
componentPositionOrientation	true false	When set to <i>true</i> , checks that the position and orientation for each assembly node in a JT file

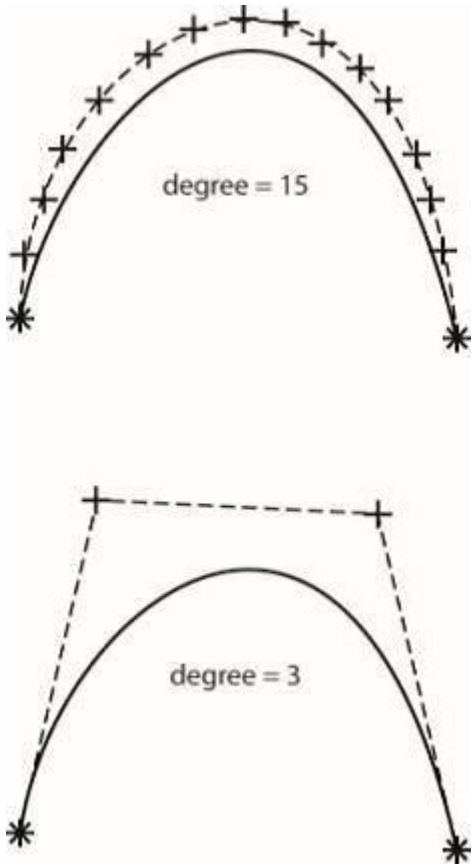
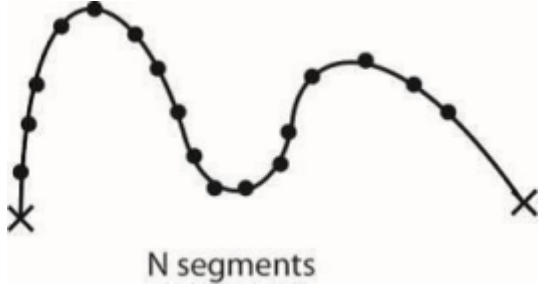
Option	Keyword or value	Description
		matches the position and orientation of the corresponding node in the source CAD file.
Close brace	}	}

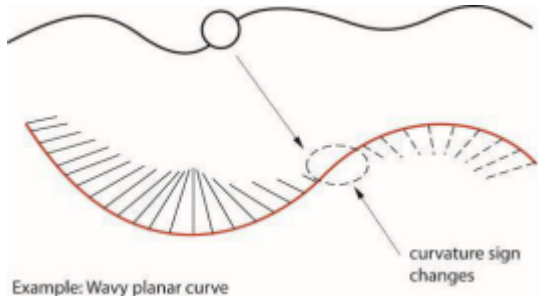
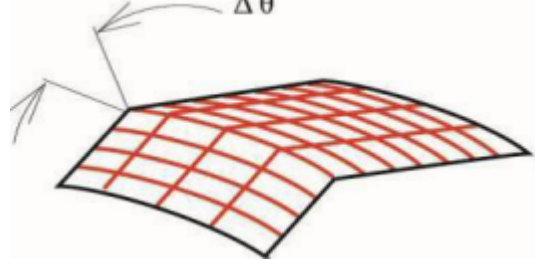
Edit SASIGIntegrity options

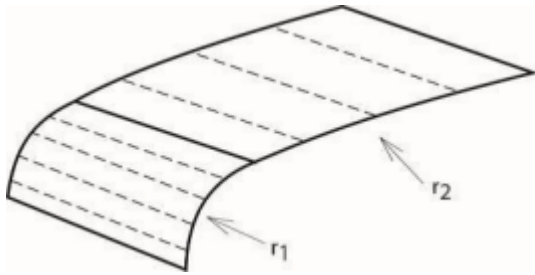
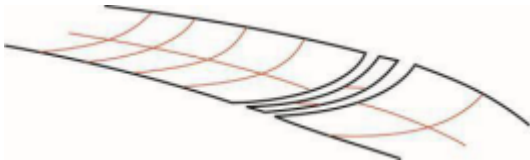
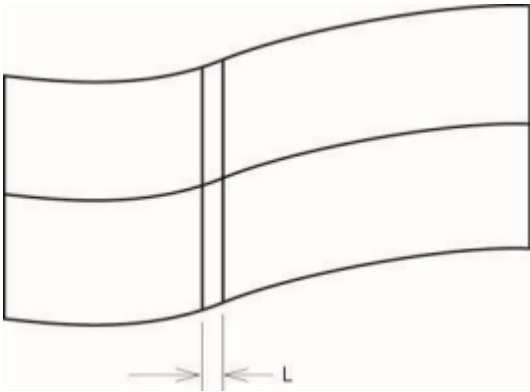
Option	Description
SASIGIntegrity {	
Curve Criteria	
largeCurveGap(G-CU-LG) = true distanceTol(G-CU-LG) = "0.001 millimeters"	G-CU-LG- Large segment gap  Definition: The distance between end points of adjacent curve or line segments is excessive. Curves may be separated by a gap or may overlap. Measurement: gap width or overlap in mm Threshold: $\geq 0.001\text{mm}$
nonTangentCurves(G-CU-NT) = true angleTol(G-CU-NT) = "3.0 degrees"	G-CU-NT-Non-tangent curves(G1 discontinuity)  Definition: Tangent angle changes at the common endpoint of adjacent curves, $\Delta\theta (\leq 90^\circ)$. Measurement: difference in tangent angle of two curves at common end point, $\Delta\theta$
nonSmoothCurves(G-CU-NS) = true	G-CU-NS – Non-smooth curves (G2 discontinuity)

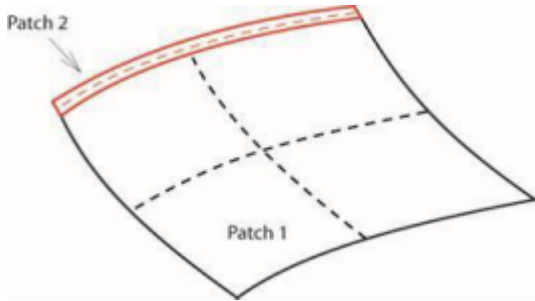
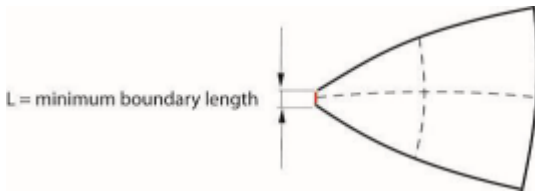
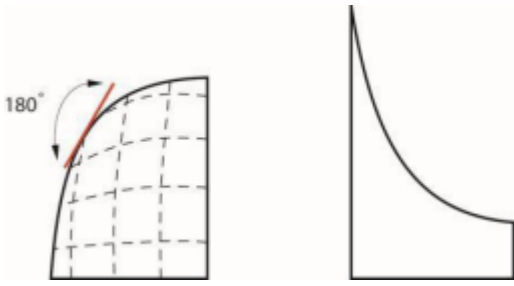
Option	Description
curvatureTol(G-CU-NS) = "0.1"	 Definition: Radius of curvature changes at the common end point of two curves or a curve and a line segment. Measurement: ratio of the difference in radius of curvature over mean radius, $r1-r2 /(r1+r2)/2$
tinyCurveOrSegment(G-CU-TI) = true curveLengthTol(G-CU-TI) = "0.01 millimeters"	G-CU-TI – Tiny Curve  Definition: Curve or arc length is smaller than permissible minimum. Measurement: segment length or arc length in mm.
indistinctCurveKnots(G-CU-IK) = true knotTolerance(G-CU-IK) = "0.00001"	G-CU-IK – Indistinct curve knots  Definition: Two or more knot points on a spline curve positioned too close to each other. Measurement: distance between knot points, Δt
selfIntersectingCurve(G-CU-IS) = true	G-CU-IS – Self-intersecting curve

Option	Description
	 Definition: A curve intersects itself at one or more locations. Self-intersections are usually caused by errors in creating offset curves. Within threshold value, the logical value is true if there is no intersection.
embeddedCurves(G-CU-EM) = true identicalCurveDistanceTol(G-CU-EM) = "0.01 millimeters"	G-CU-EM-Embedded curves  Description: A set of overlapping curves of any type. Embedded curves may be caused by CAD operator error (such as inserting the same curve twice) or by copying geometry from another source into the model. Within precision of 0.01 mm, the logical value is true if there is no embedded curve.
excessivelyHighDegreeCurve(G-CU-HD) = true maxCurveDegree(G-CU-HD) = "5"	G-CU-HD-Excessively high-degree curve

Option	Description
	 Definition: Polynomial curve is of excessively high degree, integer value n.
fragmentedCurve(G-CU-FG) = true maxSegmentCount(G-CU-FG) = "10"	G-CU-FG – Fragmented curve  N segments Definition: The curve contains too many segments, integer value N. This condition may be caused by data translation that approximates a higher degree curve with a set of lower-degree curves.
wavyPlanarCurve(G-CU-WV) = true maxWaveCount(G-CU-WV) = "3"	G-CU-WV – Wavy planar curve

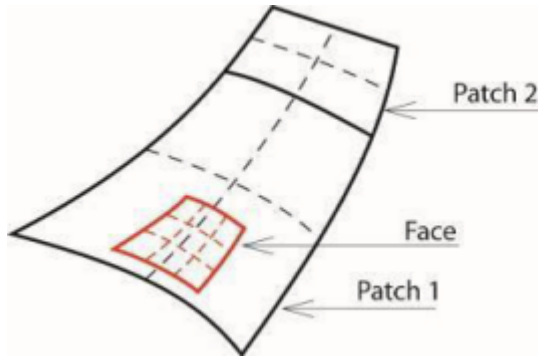
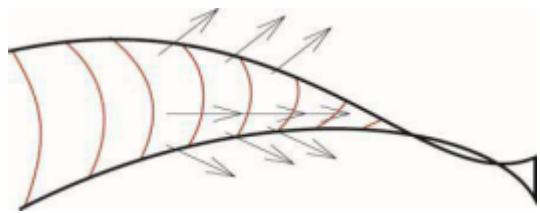
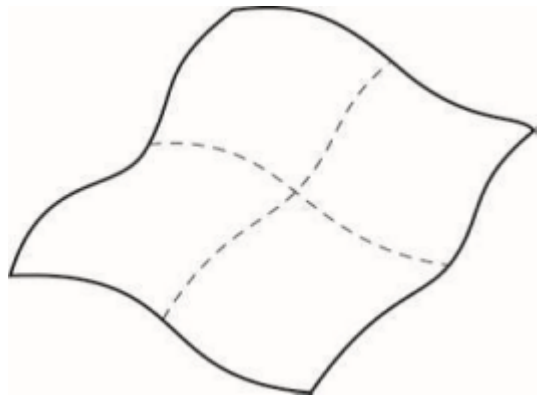
Option	Description
	 Definition: Curvature changes direction excessively. Number of changes in curvature sign = N. The condition may occur in planar or three-dimensional curves.
smallRadiusOfCurvature(G-CU-CR) = true minimumCurvatureRadius(G-CU-CR) = "0.05 millimeters"	C-CU-CR – Small radius of curvature  Definition: Radius of curvature is below acceptable limits.
Surface Criteria	
nonTangentSurface(G-SU-NT) = true angleTol(G-SU-NT) = "3.0 degrees"	G-SU-NT – Non-tangent surfaces (G1 discontinuity)  Definition: Tangent angle changes at the common boundary between adjacent surfaces or patches. Measurement: angle between normal vectors at the common boundary, $\Delta\theta$.
nonSmoothSurfacesOrPatches(G-SU-NS) = true curvatureTol(G-SU-NS) = "0.1"	G-SU-NS – Non-smooth surfaces (G2 discontinuity)

Option	Description
	 Definition: Radius of curvature changes abruptly in magnitude at the common boundary of two surfaces or patches. Measurement: ratio of the difference in radius of curvature over mean radius, $r_1 - r_2 / (r_1 + r_2) / 2$
tinySurfacePatch(G-SU-TI) = true surfacePatchTol(G-SU-TI) = "0.00001 millimeters"	G-SU-TI – Tiny surface  Definition: Area of surface or patch is below permissible minimum.
narrowSurfaceOrPatch(G-SU-NA) = true distanceTol(G-SU-NA) = "0.005 millimeters"	G-SU-NA – Narrow surface  Definition: Width of surface or patch, L, is below permissible minimum in any direction.
relativelyNarrowNeighboringPatches(G-SU-RN) = true relativeTolerance(G-SU-RN) = "0.01"	G-SU-RN – Relatively narrow

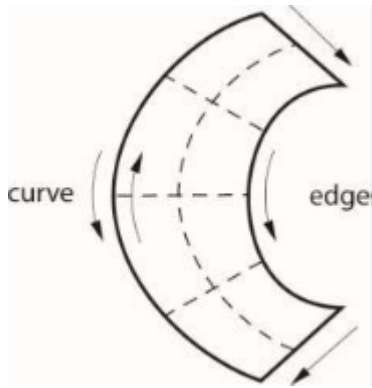
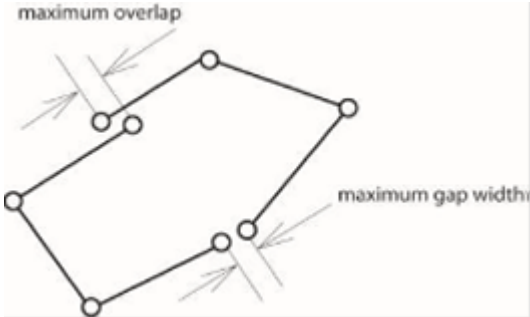
Option	Description
	 Definition: Ratio of width of surface or patch compared with adjacent patches, r, is below permissible minimum even though the absolute patch dimensions exceed allowable minimums. Measurement: $r = \text{width}(\text{patch 2}) / \text{width}(\text{patch 1})$, $r \leq 1.0$
<code>degenerateSurfaceBoundary(G-SU-DC) = true</code> <code>distanceTol(G-SU-DC) = "0.01 millimeters"</code>	G-SU-DC – Degenerate surface boundary  $L = \text{minimum boundary length}$ Definition: Length of the edge of one surface or patch boundary, L, is below permissible minimum. Some CAD systems define a triangular surface by setting one boundary below a specified limit.
<code>degenerateSurfaceCorner(G-SU-DP) = true</code> <code>cornerAngleTolerance(G-SU-DP) = "10.0 degrees"</code>	G-SU-DP – Degenerate surface corner  Definition: One or more corners of a surface meet at a sharp or tangent angle below acceptable minimums. Measurement: angle between tangents of adjacent surface boundaries at the common endpoint, $\Delta\theta$
<code>indistinctSurfaceKnots(G-SU-IK) = true</code>	G-SU-IK – Indistinct surface knots

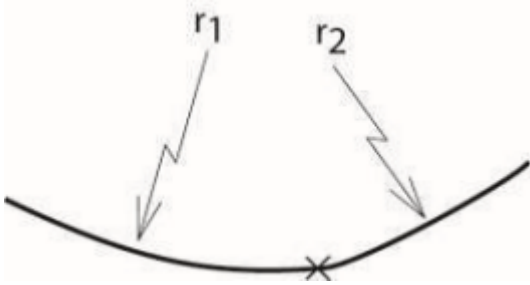
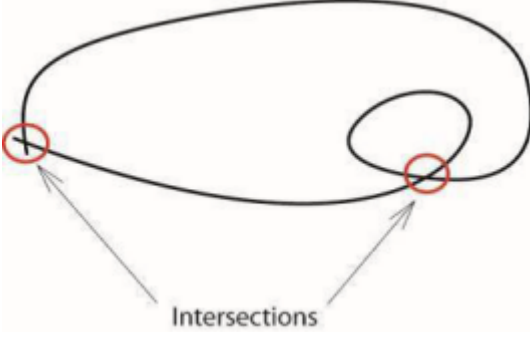
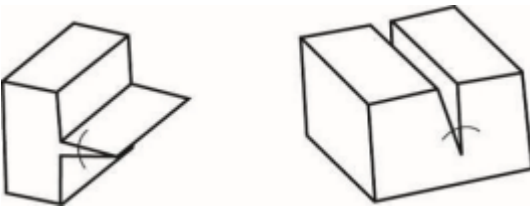
Option	Description
knotTol(G-SU-IK) = "0.000001"	Definition: Two or more knot points on a spline curve positioned too close to each other. Measurement: distance between knot points, Δu and Δv
selfIntersectingSurface(G-SU-IS) = true	G-SU-IS – Self-intersecting surface Definition: A surface or patch intersects itself at one or more locations. Within the threshold value, the logical value is true if there is no intersection.
embeddedSurfaces(G-SU-EM) = true maxSurfaceCount(G-SU-EM) = "2"	G-SU-EM – Embedded surfaces

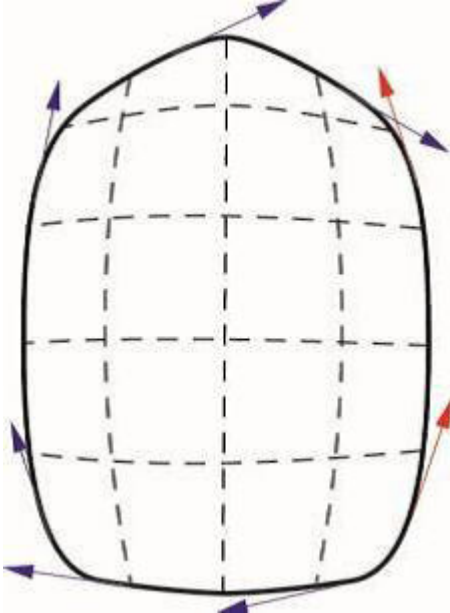
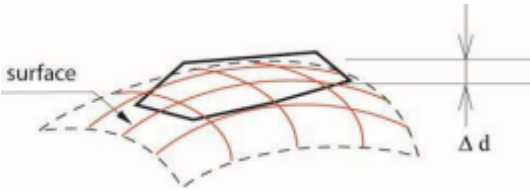
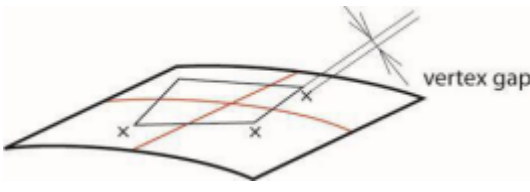
Option	Description
	Description: A set of completely overlapping surfaces of any type. Within precision of 0.01 mm, the logical value is true if there is no embedded surface.
excessivelyHighDegreeSurface(G-SU-HD) = true maximumDegree(G-SU-HD) = "5"	G-SU-HD – Excessively high-degree surface Definition: Polynomial surface is of excessively high degree.
fragmentedSurface(G-SU-FG) = true maxPatchCount(G-SU-FG) = "2500"	G-SU-FG – Fragmented surface Definition: Number of patches in the surfaces, N, exceeds recommended value. The condition may be caused by translation processes that approximate high-degree polynomial surfaces with surfaces of lower degree or by the merging of surface elements with varying segmentation.
unusedPatches(G-SU-UN) = true	G-SU-UN – Unused patches

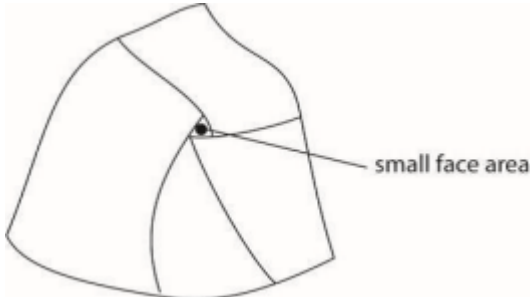
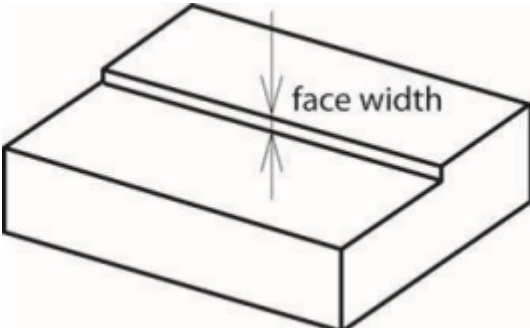
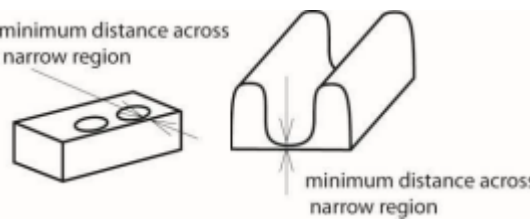
Option	Description
	 Definition: Surface entity contains patches that are not used in any face of a solid. Unused patches are generally produced when the bounded face of a solid is much smaller than the surface defining the face.
foldedSurface(G-SU-FO) = true flappingAngleTolerance(G-SU-FO) = "90.0 degrees"	G-SU-FO – Folded surface 
wavySurface(G-SU-WV) = true maxCurvatureSignChanges(G-SU-WV) = "3"	G-SU-WV – Wavy surface  Definition: The maximum angle between pairs of normal vectors, θ, in either parametric direction of a patch or surfaces exceeds the allowable limit. Generally normal vectors of a surface uniformly face in nearly the same direction, either into the solid or out of it.
smallSurfaceRadiusOfCurvature(G-SU-CR) = true	G-SU-CR – Small surface Radius of curvature

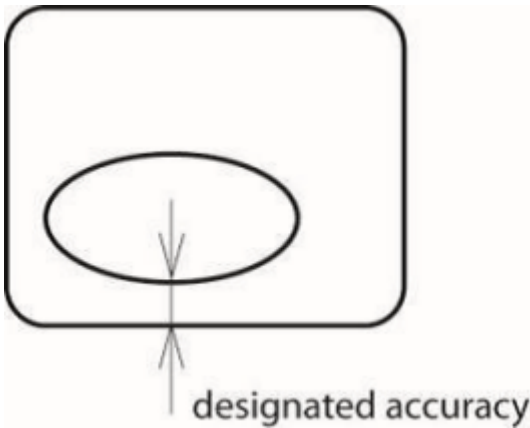
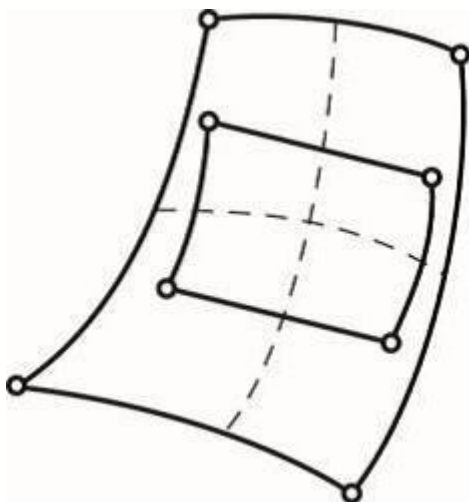
Option	Description
$\text{minimumCurvatureRadius}(\text{G-SU-CR}) = \text{"0.05 millimeters"}$	A diagram showing a curved surface with a red line segment. An arrow labeled 'R' points to the radius of curvature of the surface. Definition: Radius of curvature, R, within the surface is below the minimum allowable value.
Edge Criteria	
$\text{tinyEdge}(\text{G-ED-TI}) = \text{true}$ $\text{distanceTol}(\text{G-ED-TI}) = \text{"0.01 millimeters"}$	G-ED-TI – Tiny edge A diagram showing a curved surface with a red line segment. A double-headed arrow labeled 'minimum edge length' indicates the length of the segment. Definition: Length of an edge curve or line segment falls below acceptable minimum.
$\text{fragmentedEdge}(\text{G-ED-FG}) = \text{true}$ $\text{maxSegmentCount}(\text{G-ED-FG}) = \text{"10"}$	G-ED-FG – Fragmented edge A diagram showing a curved surface with a red line segment. A dashed line indicates the maximum segment count of the edge. Definition: The number of segments in an edge curve exceeds the allowable maximum.
$\text{inconsistentEdgeOnCurve}(\text{G-ED-IT}) = \text{true}$	G-ED-IT – Inconsistent edge on curve

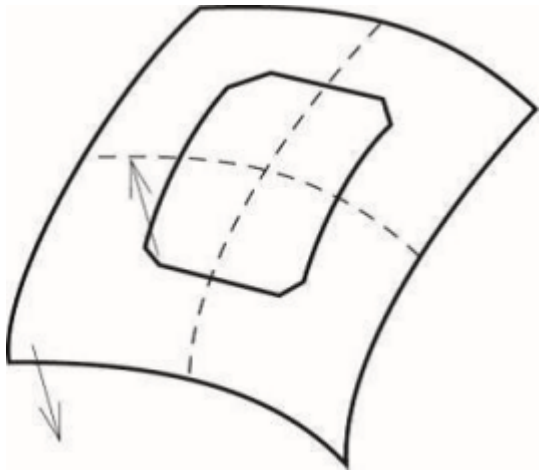
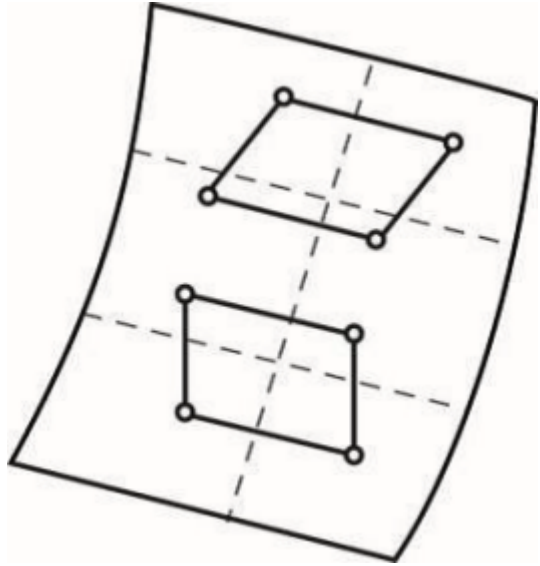
Option	Description
	 Definition: The direction of an edge curve in parameter space is inconsistent with its underlying curve in 3D space. The criterion is true if the directions are consistent.
Edge Loop Criteria	
largeEdgeGap(G-LO-LG) = true edgeGapTol(G-LO-LG) = "1.0 millimeters"	G-LO-LG – Large edge gap  Definition: Distance between end points of adjacent edges is excessive (G0 discontinuity). Curves may be separated by a gap or may overlap.
nonTangentEdges(G-LO-NT) = true angleTol(G-LO-NT) = "3.0 degrees"	G-LO-NT – Nontangent edges  Definition: Tangent angle changes at the common endpoint of adjacent edges, $\Delta\theta (\leq 90^\circ)$ (G1 discontinuity).
nonSmoothEdges(G-LO-NS) = true curvatureTol(G-LO-NS) = "0.1"	G-LO-NS – Non-smooth edges

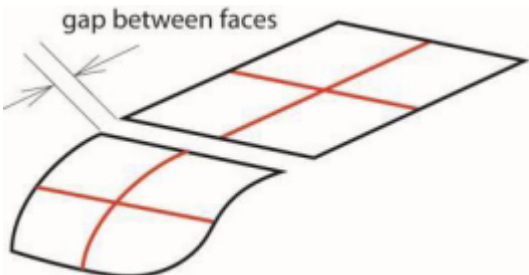
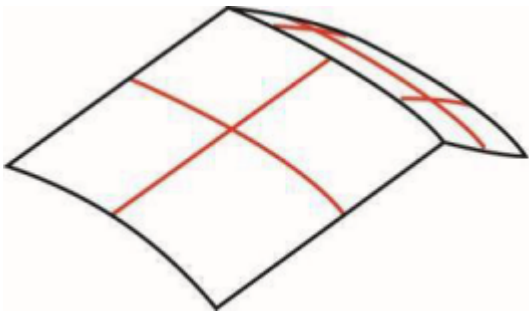
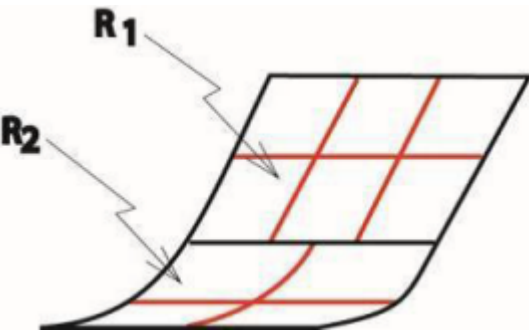
Option	Description
	 Definition: Radius of curvature changes at the common end point of two edge segments.
selfIntersectingLoop(G-LO-IS) = true	G-LO-IS – self intersecting loop  Definition: An edge loop intersects itself at one or more locations. Self-intersections may be caused by errors in creating offset entities. Criterion is true if edge loops do not intersect themselves.
sharpEdgeAngle(G-LO-SA) = true angleTol(G-LO-SA) = "0.5 degrees"	G-LO-SA – Sharp edge Angle  Definition: Segments in an edge loop meet at angles below the recommended minimum.
inconsistentEdgeInLoop(G-LO-IT) = true	G-LO-IT – Inconsistent edge in loop

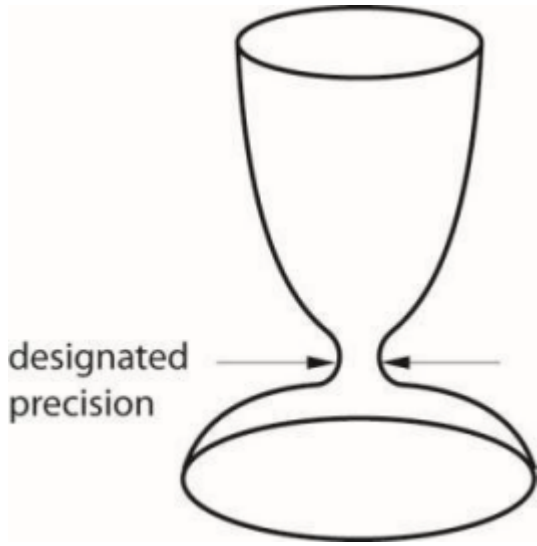
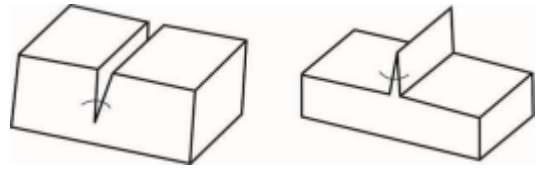
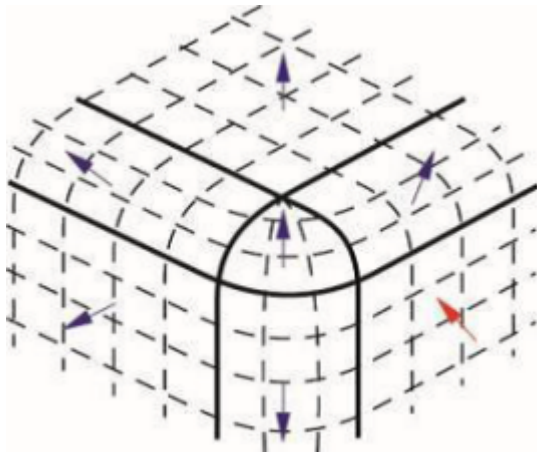
Option	Description
	 Definition: The direction of an edge loop in parameter space is inconsistent with its underlying boundary curve. The criterion is true if the directions are consistent.
Face Criteria	
largeEdgeFaceGap(G-FA-EG) = true faceGapTol(G-FA-EG) = "1.0 millimeters"	G-FA-EG – Large edge face gap  Definition: The distance between any point on an edge and the closest point on the base surface exceeds the allowable maximum.
largeVertexGap(G-FA-VG) = true vertexGapTol(G-FA-VG) = "1.0 millimeters"	G-FA-VG – Large vertex gap  Definition: The distance between a vertex and the corresponding edge end point or face that it trims exceeds the allowable maximum.

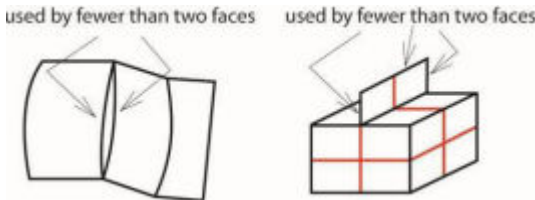
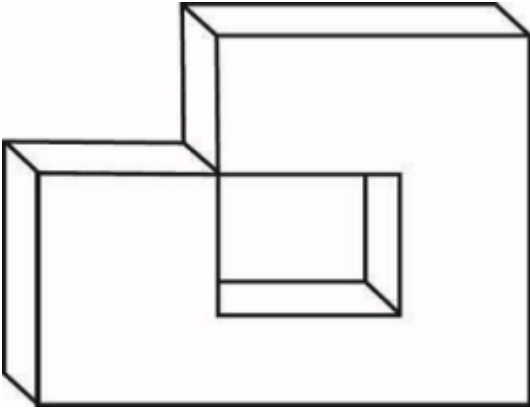
Option	Description
tinyFace(G-FA-TI) = true maxAreaSize(G-FA-TI) = "0.00001 millimeters"	G-FA-TI – Tiny Face  small face area Definition: Area of a face is below the allowable minimum.
narrowFace(G-FA-NA) = true narrowFaceTol(G-FA-NA) = "0.01 millimeters"	G-FA-NA – Narrow face  face width Definition: Width of a face in one direction is consistently below the allowable minimum.
narrowRegion(G-FA-RN) = true narrowRegionTol(G-FA-RN) = "0.01 millimeters"	G-FA-RN – Narrow region  minimum distance across narrow region minimum distance across narrow region Definition: The distance between the two closest points in a loop or two loops in the same face falls below the allowable minimum.
intersectingLoop(G-FA-IS) = true	G-FA-IS – Intersecting loops

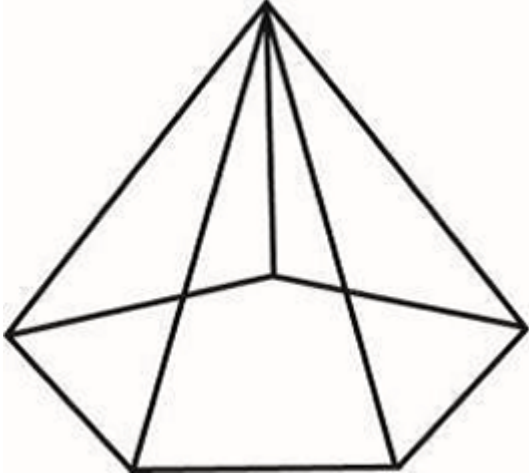
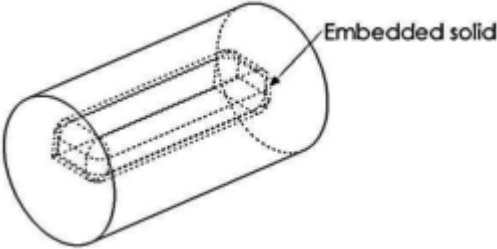
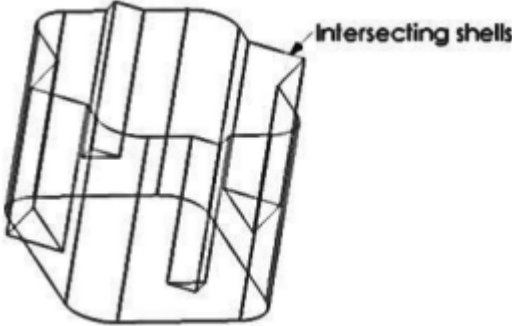
Option	Description
	 Definition: Two or more loops in the same face intersect one another. Intersecting loops may render the face invalid. The integrity of the model topology may be compromised. The criterion is true if the faces are not completely embedded within allowable system tolerances.
embeddedFace(G-FA-EM) = true embeddedFaceTol(G-FA-EM) = "0.01 millimeters"	G-FA-EM – Embedded faces  Description: A set of completely overlapping faces of any type. The criterion is true if the faces are not completely embedded within allowable threshold value
inconsistentFaceOnSurface(G-FA-IT) = true	G-FA-IT – Inconsistent face on surface

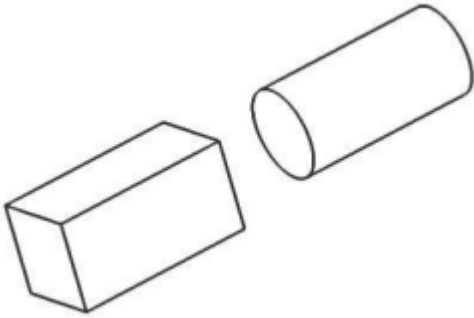
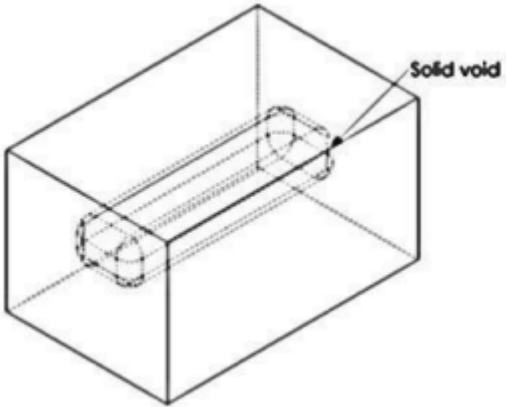
Option	Description
	 Definition: The direction of the face normal vectors is inconsistent with its underlying surface. The criterion is true if the directions are consistent. Some CAD systems don't assign a normal orientation to faces, so faces and their associated surfaces are always consistent. The criterion is true if the face and base surface normal vectors are consistent.
multiRegionFace(G-FA-MU) = true	G-FA-MU – Multi región surface  Definition: The number of faces, N, associated with a single base surface.
Shell Criteria:	
largeFaceGap(G-SH-LG) = true faceGapTol(G-SH-LG) = "1.0 millimeters"	G-SH-LG – Large face gap

Option	Description
	 Definition: Excessive distance between the boundaries of adjacent faces (G0 discontinuity)
nonTangentFaces(G-SH-NT) = true angleTol(G-SH-NT) = "0.5 degrees"	G-SH-NT – Non-tangent faces  Definition: Adjacent faces are not tangent (G1 discontinuity).
nonSmoothFaces(G-SH-NS) = true curvatureTol(G-SH-NS) = "0.1"	G-SH-NS – Non-smooth faces  Definition: Radius of curvature changes at the common boundary of two faces (G2 discontinuity).
selfIntersectingShell(G-SH-IS) = true	G-SH-IS – self-intersecting Shell

Option	Description
	 Definition: A surface or patch intersects itself at one or more locations. The criterion is true if the shell does not intersect itself within specified precision.
sharpFaceAngle(G-SH-SA) = true angleTol(G-SH-SA) = "5.0 degrees"	G-SH-SA – Sharp face angle  Definition: Shell faces meet at angles below the recommended minimum.
inconsistentFaceInShell(G-SH-IT) = true	G-SH-IT – Inconsistent face in Shell 

Option	Description
$\text{freeEdge}(\text{G-SH-FR}) = \text{true}$	Definition: Faces with normal vectors pointing in opposite directions share a common boundary. Or the shell normal vector orientation is opposite from the face normal vector. The criterion is true if no inconsistencies exist. G-SH-FR – Free edge  Definition: An edge curve is used by only one face in a shell. The criterion is true if no edges are used by fewer than two faces.
$\text{overUsedEdge}(\text{G-SH-NM}) = \text{true}$ $\text{maxEdgeCount}(\text{G-SH-NM}) = "2"$	G-SH-NM – Over-used edge  Definition: An edge curve is shared by more than two faces in a shell. The criterion is true if no edges are used by more than two faces.
$\text{overUsedVertex}(\text{G-SH-OU}) = \text{true}$ $\text{maxEdgeCount}(\text{G-SH-OU}) = "3"$	G-SH-OU – over-used vertex

Option	Description
	 Definition: A vertex is shared by too many edges. The criterion value is an integer, N, equal to the number of edges shared by a vertex.
Solid body criteria: <code>embeddedSolids(G-SO-EM) = true</code> <code>maxSolidCount(G-SO-EM) = "2"</code>	G-SO-EM – Embedded solids  Description: Solid volumes completely surrounded by other solid volumes. The logical value of the criterion is true if no solids are embedded within other solids.
<code>intersectingShells(G-SO-IS) = true</code>	G-SO-IS – Intersecting shells 

Option	Description
	Definition: A pair of shells intersect one another in a solid body. The logical value of this criterion is true if no shells intersect within the solid within the specified limits of precision.
multiVolumeSolid(G-SO-MU) = true	G-SO-MU – Multi-volume solid  Description: A model contains multiple solid bodies that do not intersect.
tinySolid(G-SO-TI) = true volTolerance(G-SO-TI) = "0.01 millimeters"	G-SO-TI – Tiny solid Definition: The overall extent of the solid is the below permissible minimum.
solidVoid(G-SO-VO) = true	G-SO-VO – Solid void  Description: Internal or external void in a solid model. The logical value of the criterion is true if no internal cavity exists.

Working with JT Inspector results

Overview of working with JT Inspector results

JT Inspector test results appear on the **Results** tab of the **Inspector** dialog box. There is a Summary of all test results and Details for each inspected part and assembly. Icons help you identify successful tests, failures, and information.

On the **Results** tab, you can:

- View a summary of the inspection results and details for each part.
- Expand and collapse report sections by selecting the plus or minus symbols.
- Highlight inspected parts in the Viewing window.
- Filter the results by selecting **All**, **Pass**, **Fail**, **Warning**, or **Info**.
- View failures in the Viewing window.
- View part entity IDs.
- Highlight part entities in the Viewing window.
- Determine if a JT file contains B-Rep data.
- Determine if parts have B-Rep Entity IDs and view any face and edge IDs.
- Save Inspection results to an XML file.

View part entity IDs

If your JT file contains B-Rep data with Entity IDs, you can view the IDs and zoom to the corresponding part entities.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.
3. On the **Results** tab, expand the **Details** section one level.

4. Expand a part to examine.
5. Expand **JT Content**.
6. Expand **Geometry**.
7. Expand **XT B-Rep**, **JT B-Rep**, or **ULP**.
8. Expand **Faces** and **Edges**.

You see the IDs for the face and edge entities.

9. (Optional) To highlight an entity in the Viewing window, select its ID.
10. (Optional) To zoom to the selected entity in the Viewing window, select the **Fly to Selected** check box.

Tip:

Highlighted faces and edges may be hidden by other geometry. To view hidden faces and edges, set your display mode to draw tessellation lines. In this mode, highlights are still drawn in shaded mode.

Highlight part entities in the Viewing window

If your JT file contains B-Rep data with Entity IDs, you can select an entity on the **Results** tab of the **Inspector** dialog box to highlight the entity in the Viewing window. You can also zoom to the selected entity.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.
3. On the **Results** tab, expand the **Details** section one level.
4. Expand a part to examine.
5. Expand **JT Content**.
6. Expand **Geometry**.
7. Expand **XT B-Rep**, **JT B-Rep**, or **ULP**.

8. Expand **Faces** and **Edges**.
9. To highlight a face or edge entity in the Viewing window, select its ID.

The entity is highlighted in the Viewing window.

10. To zoom to the entity, select the **Fly to Selected** check box.

Tip:

Highlighted faces and edges may be hidden by other geometry. To view hidden faces and edges, set your display mode to draw tessellation lines. In this mode, highlights are still drawn in shaded mode.

Determine if a JT file contains any B-Rep or ULP data

Determine if a JT file contains any B-Rep or Ultra-Lightweight Precise (ULP) data.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.
3. On the **Results** tab, expand the **Summary** section one level.
4. Expand **JT Content**.
5. Expand **XT B-Rep**, **JT B-Rep** and **ULP**.

If there are B-Rep or ULP data, they are listed.

Determine if a JT file contains B-Rep Entity IDs

B-Rep Entity IDs are unique identifiers for individual part entities. They are created in the CAD system. B-Rep Entity IDs are required for associative measurements.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.

3. On the **Results** tab, expand the **Details** section one level.
4. Expand the part to examine.
5. Expand **JT Content**.

If the part has no B-Rep data, you see **Geometry : No B-Rep found**.

6. If the part has B-Rep data, expand **Geometry**.
7. Expand **XT B-Rep** or **JT B-Rep**.

If the part has no B-Rep Entity IDs, you see **Entity IDs: False**.

8. If the part has B-Rep Entity IDs, expand **Faces** and **Edges**.

You see the Entity IDs for the face and edge entities.

Determine if a JT file includes STT (Smart Topology Table) data

Determine if a JT file contains any STT data.

1. Choose **Tools** tab→**Inspector** group→**JT Inspector** .

- or -

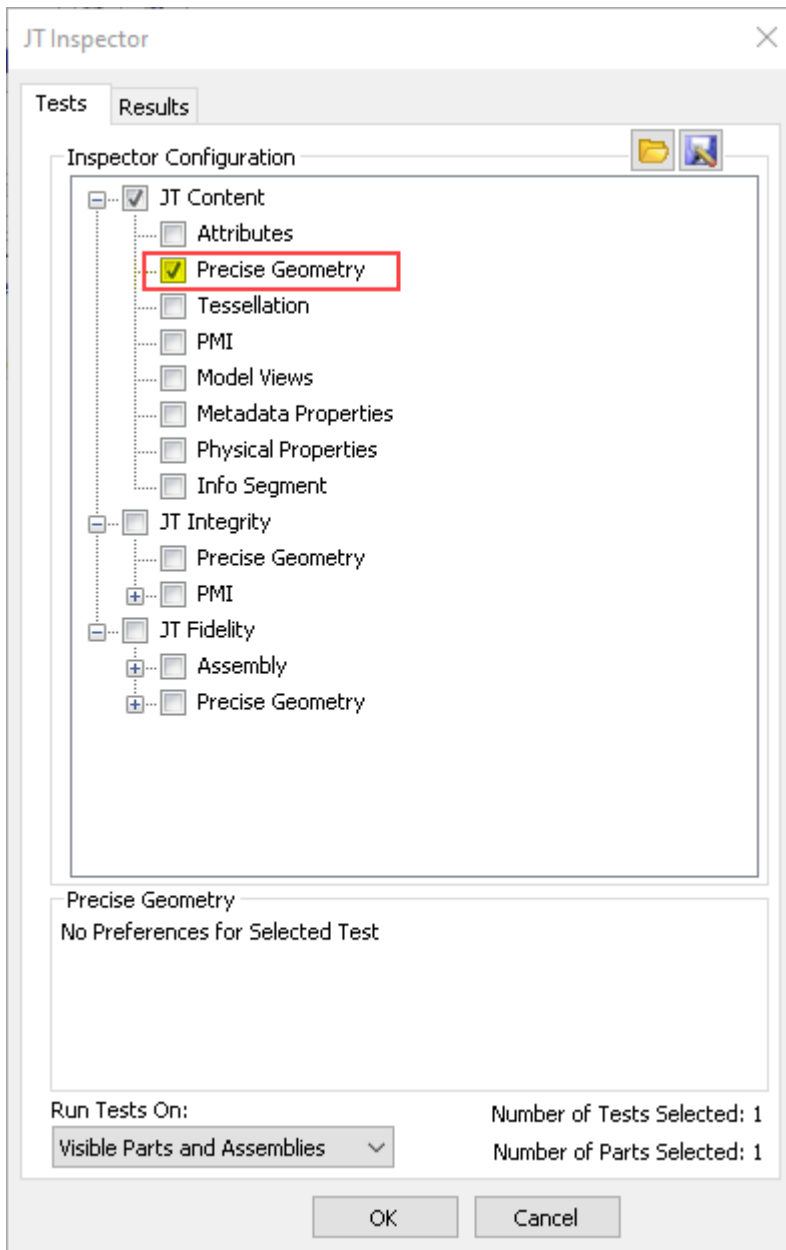
If you have the JT Inspector Design Authority license, choose **JT Inspector Design Authority** .

Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

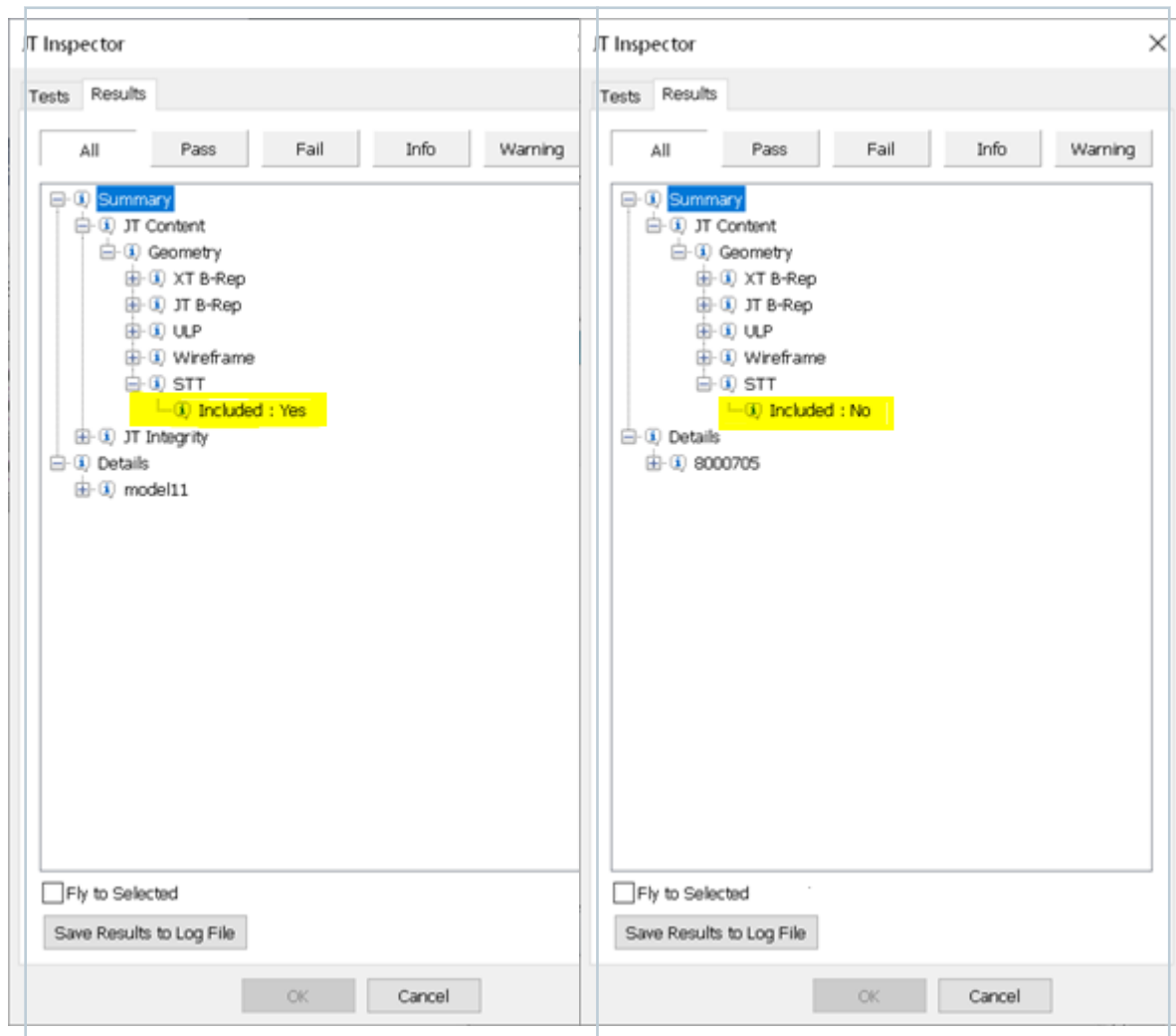
If you have the JT Inspector Design Authority license, choose **JT Inspector Design Authority**.

2. Run the **JT Content**→**Precise Geometry** tests.



3. On the **Results** tab, expand the **Summary** section one level.
4. Expand **JT Content**.
5. Expand **Geometry**.
6. Expand **STT**.

The result Yes indicates that STT is included in JT and the result No indicates that STT is not included.



Determine if a JT file has complete STT (Smart Topology Table) data

Determine if a JT file contains complete STT data. STT is complete if it fully complies with the STT definition as documented in the JT version 10.5 specification.

1. Choose **Tools** tab→**Inspector** group→**JT Inspector** .

- or -

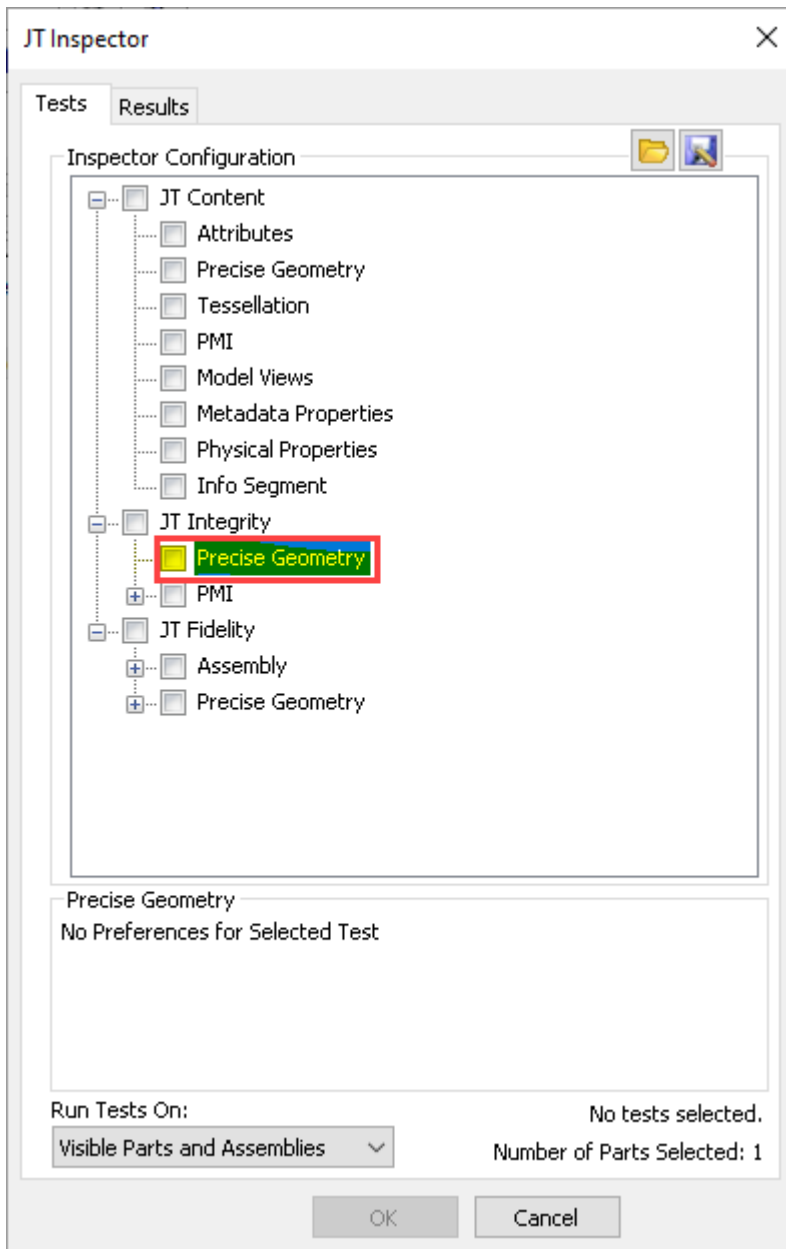
If you have the JT Inspector Design Authority license, choose **JT Inspector Design Authority** .

Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

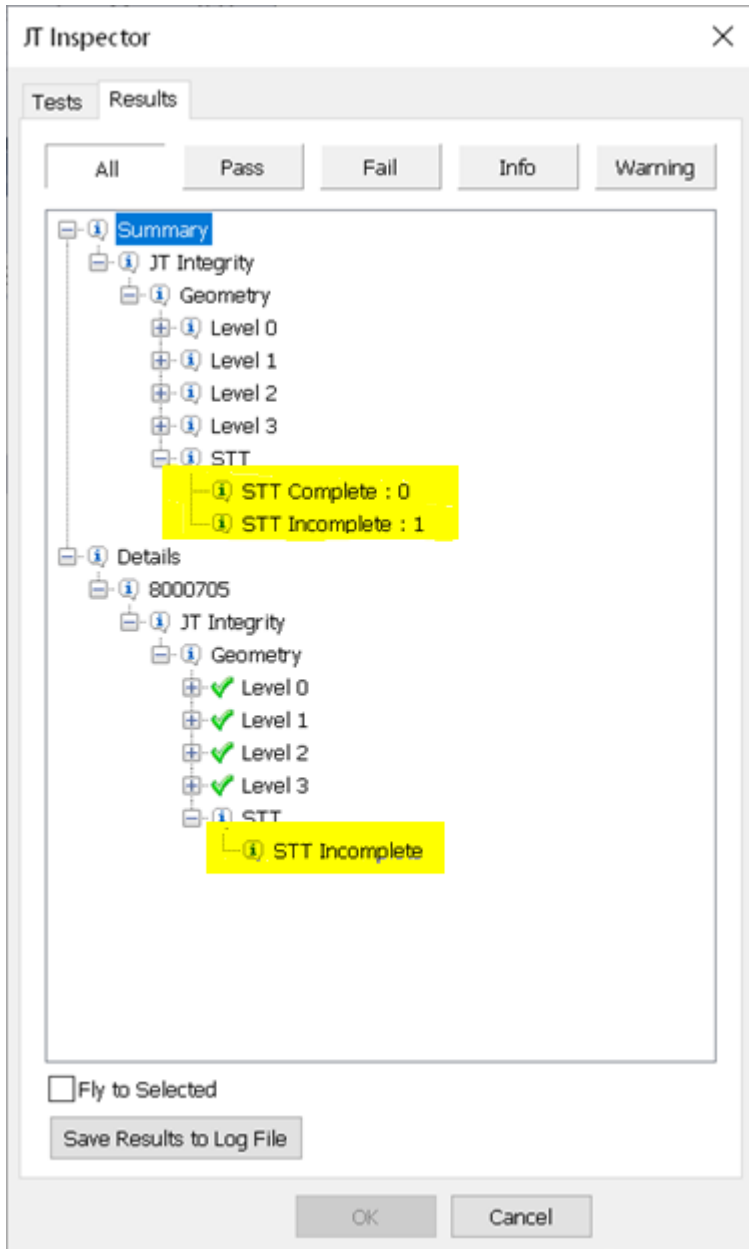
If you have the JT Inspector Design Authority license, choose **JT Inspector Design Authority**.

- Run the **JT Integrity->Precise Geometry** tests.



- On the **Results** tab, expand the **Summary** section one level.
- Expand **JT Integrity**.
- Expand **Geometry**.
- Expand **STT**.

The Summary result shows the number of parts with STT Complete or Incomplete data. As shown below, the STT Complete followed by the number indicates the number of parts with valid STT data and STT Incomplete followed by the number indicates the parts with invalid STT data. And the Details results for each part shows STT Complete or STT Incomplete.



View inspection results

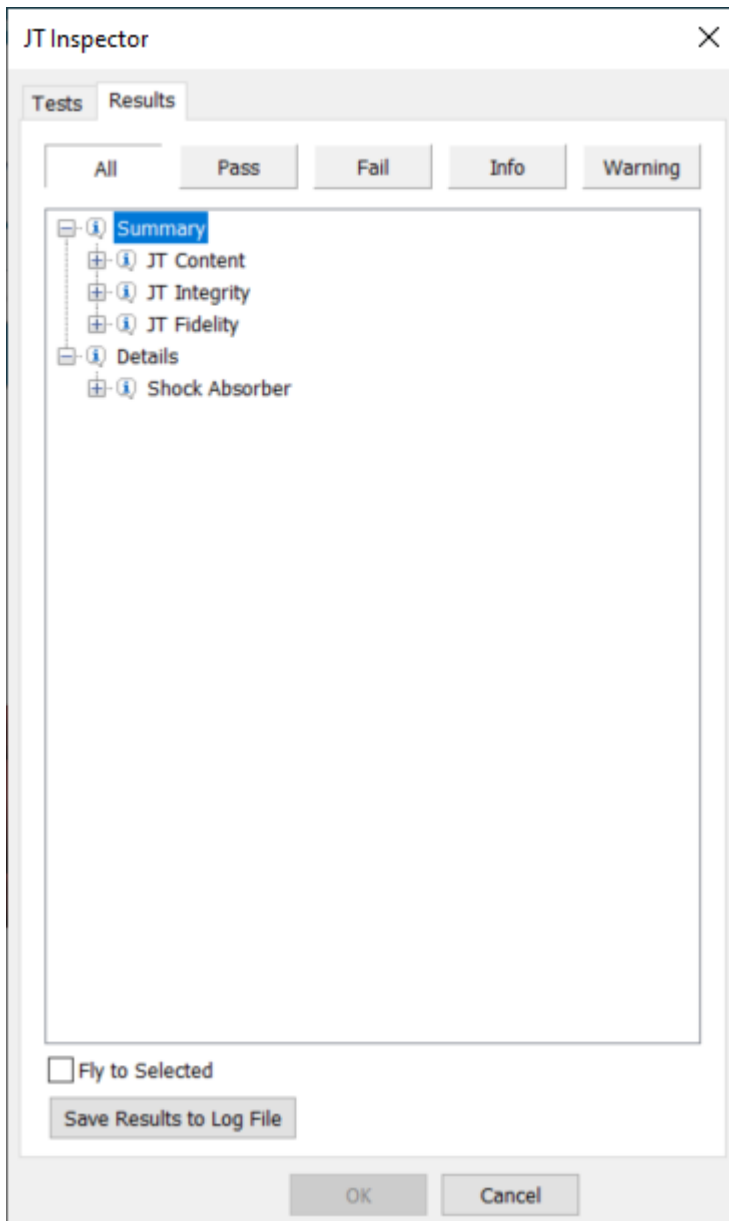
Inspection test results are displayed in two reports: Summary and Details.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.
3. On the **Results** tab, expand or collapse the Summary and Details by selecting the plus or minus symbols.



4. Use the icons to identify types of results.

This icon	Indicates
	All selected tests passed.
	One of the following: • If next to an Assembly, one of the parts within the Assembly failed. • If next to a Part, a part entity (face, edge, and so forth) failed. Note: You can expand the hierarchical assembly to explore details about the cause of the failure.
	Warning indicating that the test could not be run, for example, due to the absence of validation properties to do fidelity testing.
	Information is available for the assembly or part.

Highlight an inspected part in the Viewing window

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.
3. On the **Results** tab, expand the **Details** section one level.
4. Select a part.

The part is highlighted in the Viewing window.

Filter inspection test results

1. Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. **Run** the inspection tests.

- On the **Results** tab of the **Inspector** dialog box, select one of the following filter options:

Select	To
All	Display all inspection results.
Passed	Display results for only those parts and assemblies that passed all inspection tests.
Failed	Display results for only those parts and assemblies that failed at least one inspection test.
Warning	Display tests that could not be run.
Info	Display only the informational results for all inspected parts and assemblies.

View inspection failures in the Viewing window

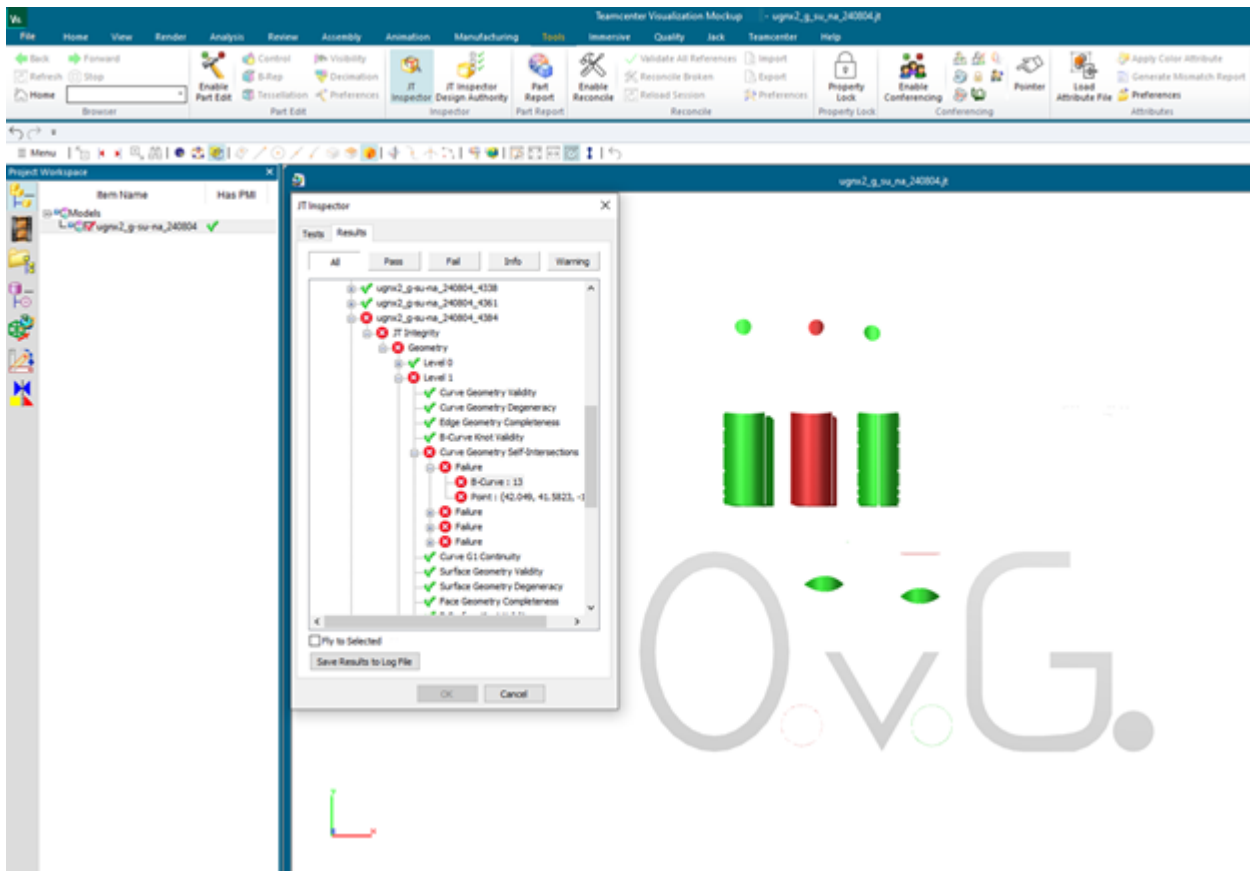
It may be easier to understand the cause of a reported failure by viewing the corresponding part failure geometry in the Viewing window.

- Choose **Actions**→**Inspector**→**JT Inspector**.

- or -

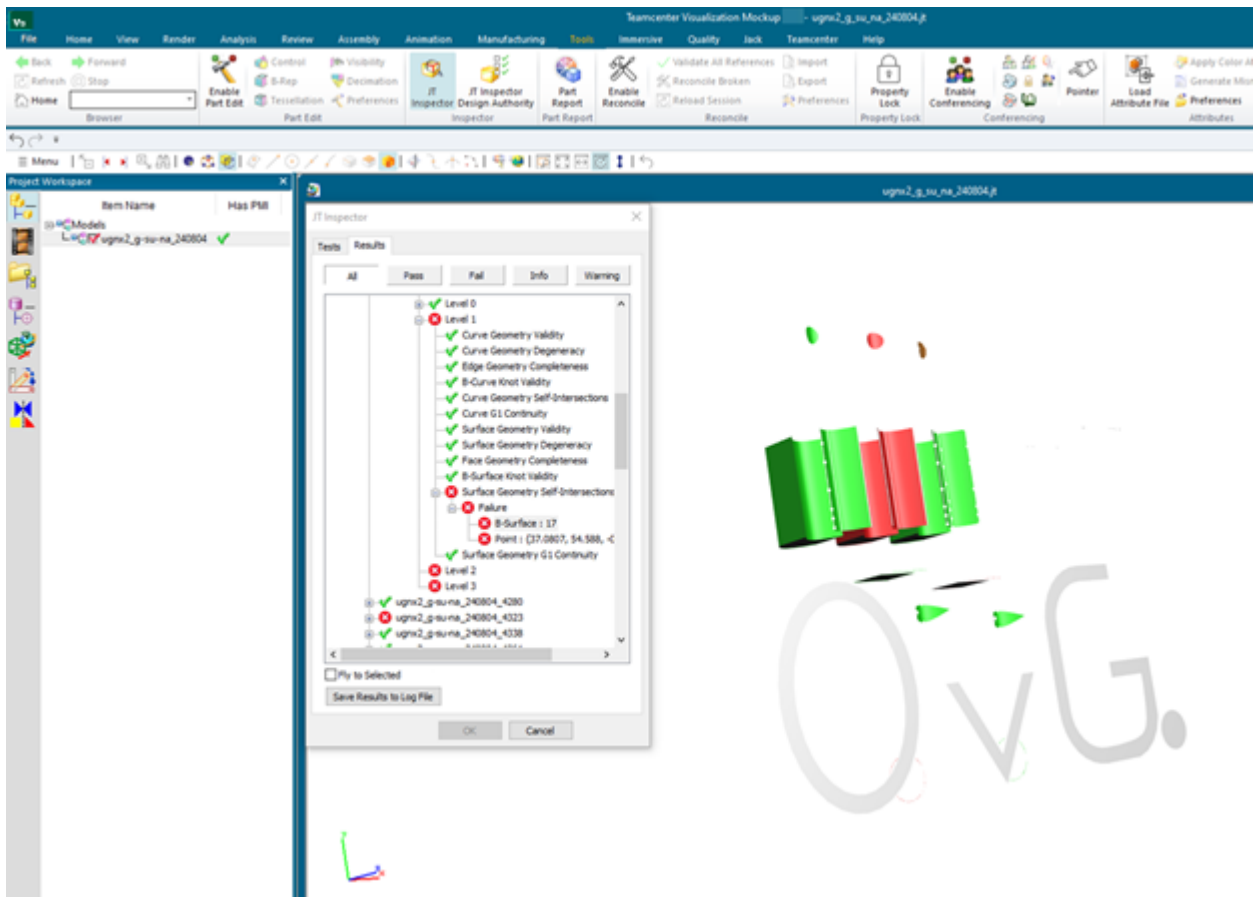
If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

- Run** the inspection tests.
- On the **Results** tab of the **Inspector** dialog box, expand the Details down to a specific failure.



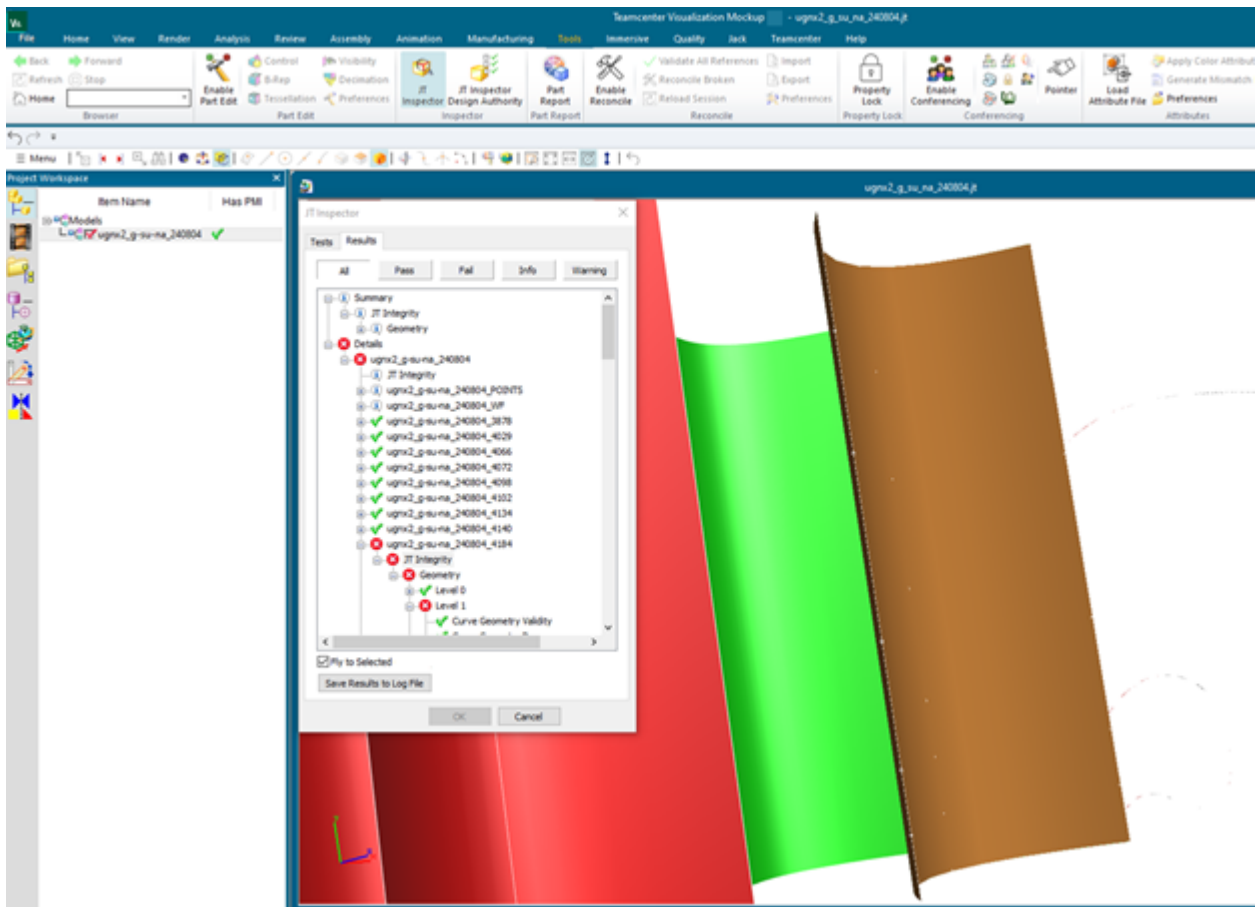
4. Select the failure item.

The part failure geometry is highlighted in the Viewing window.



5. If the failure is not visible, select the **Fly to Selected** check box.

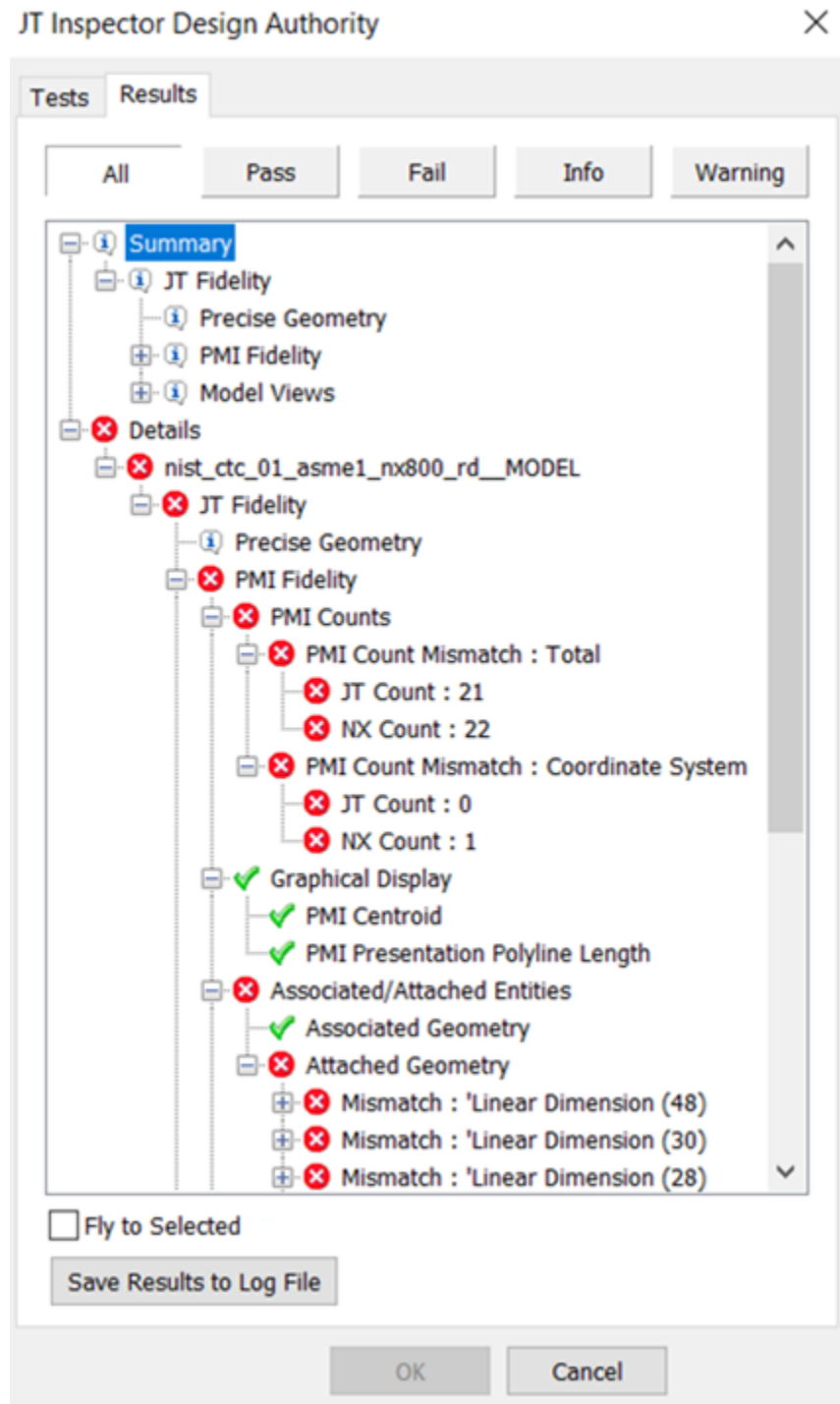
Your view zooms to the selected failure.



Reviewing PMI/Model View Inspection Results

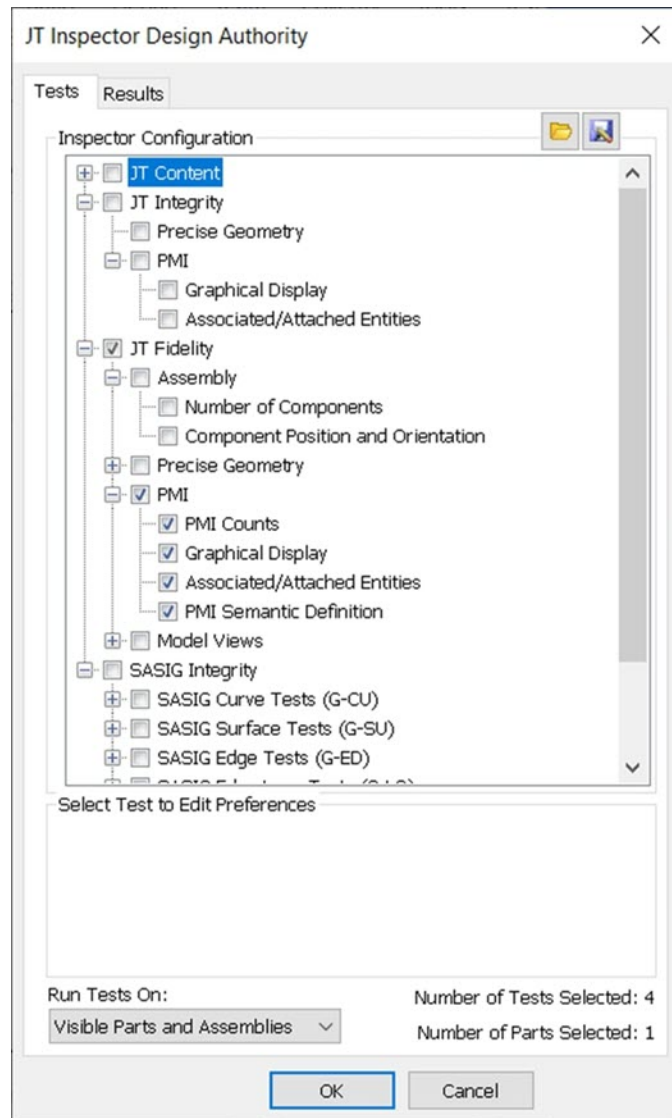
Upon running the JT Inspection, the results are displayed in the 'Results' tab of JT Inspector

Use the icons as shown below to identify types of results.

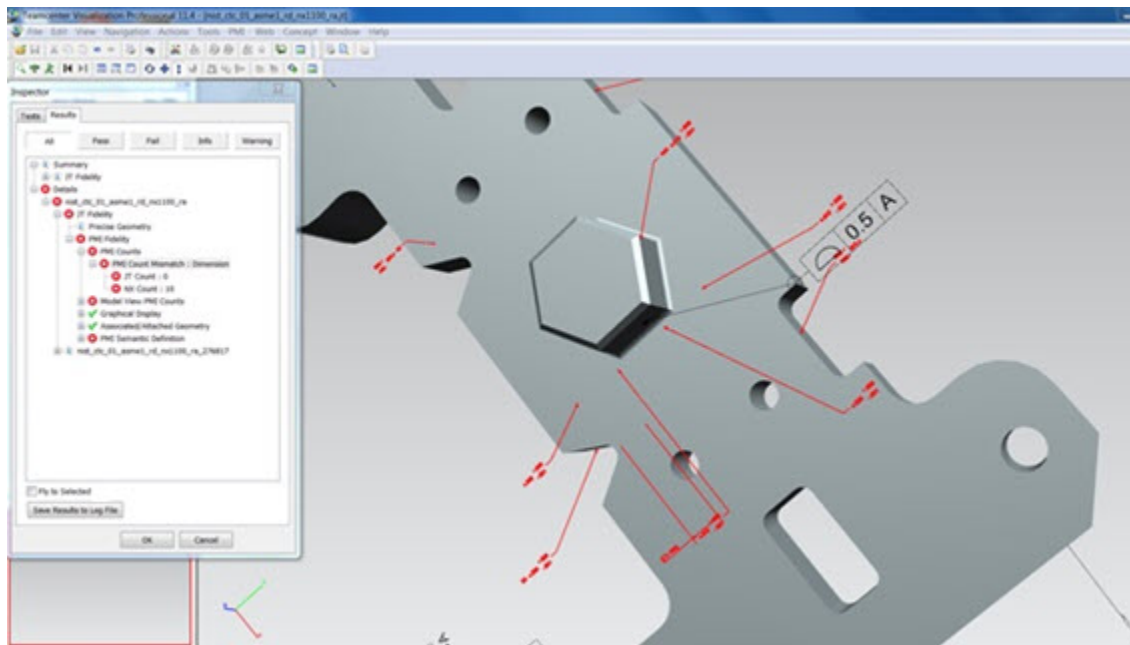


View PMI failures in the Details window

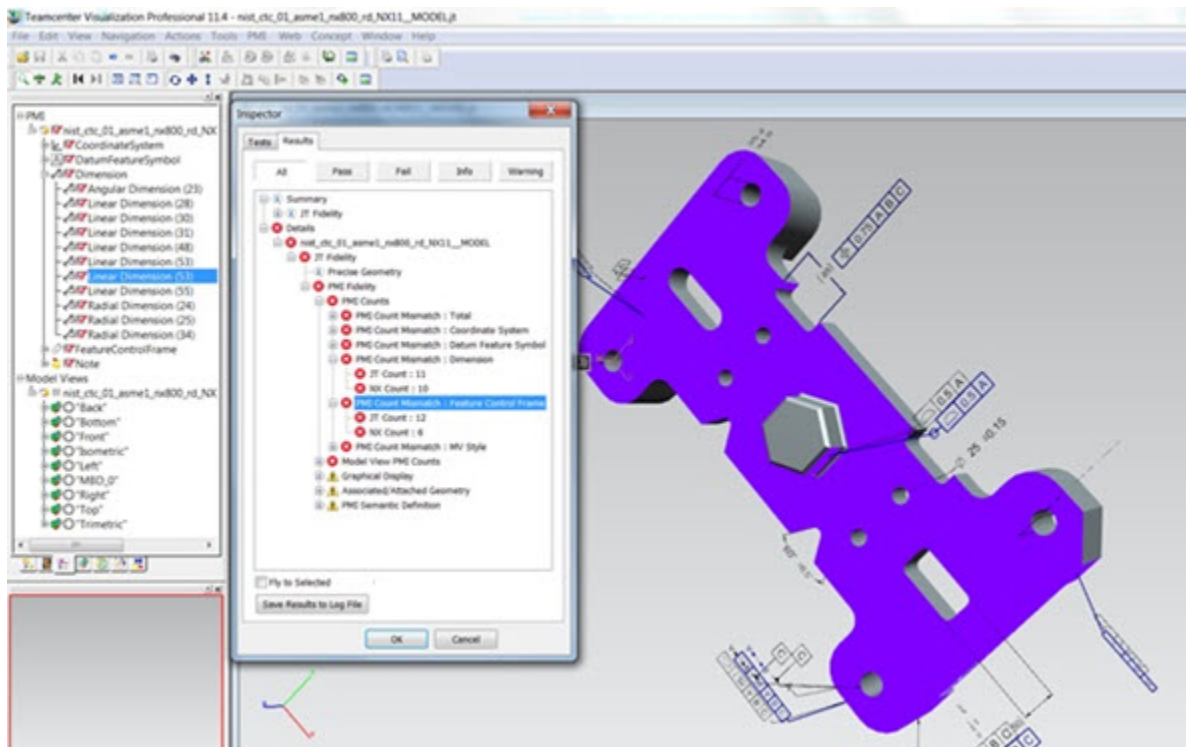
When an external fidelity source is supplied for the PMI fidelity tests, it is possible to visualize the PMI failures reported in the Details section of PMI Fidelity. Currently, the accepted format for external fidelity source is a .plmxml file.



For the test shown below, there are a number of dimensions missing in the JT as compared to the source CAD system. On selecting the PMI Count Mismatch: dimension string in the JT Inspector results windows, all the PMI that were in the source system but are missing in the JT file are rendered with red polylines.

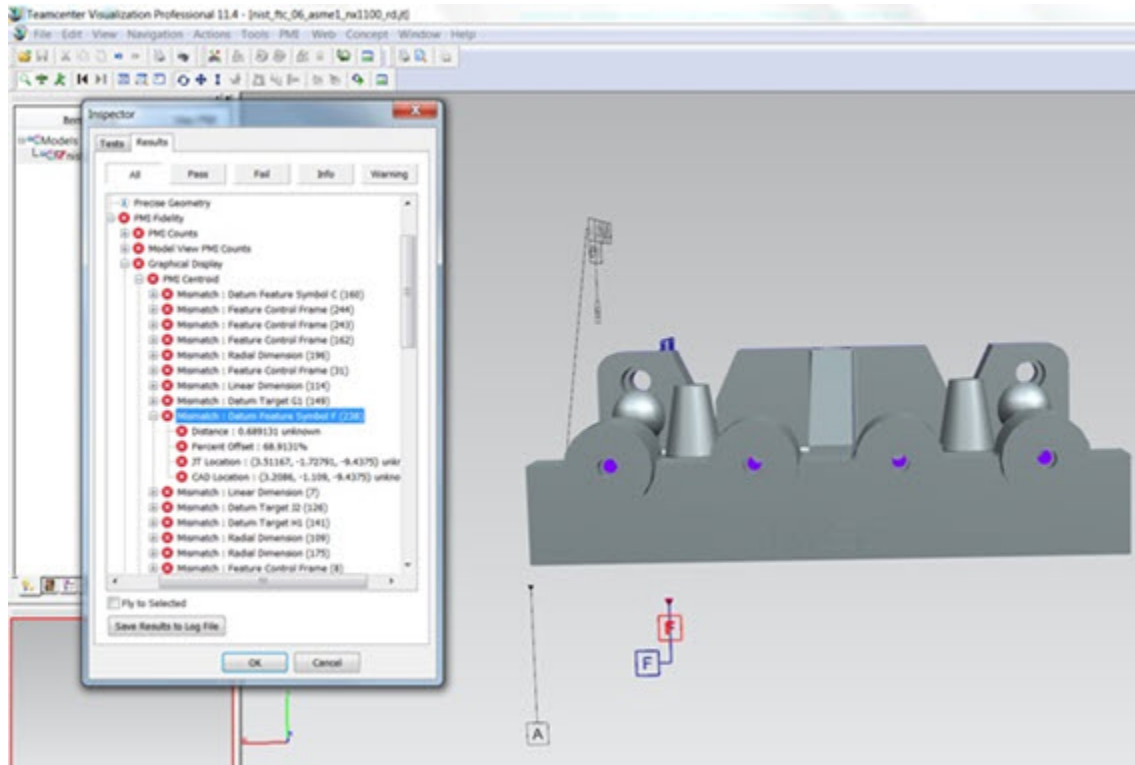


For the test shown below, there are six Feature Control Frame that are duplicated in the JT file. On selecting, PMI count mismatch: Feature Control Frame, the extra Feature Control Frames (and their associated geometry) are highlighted.

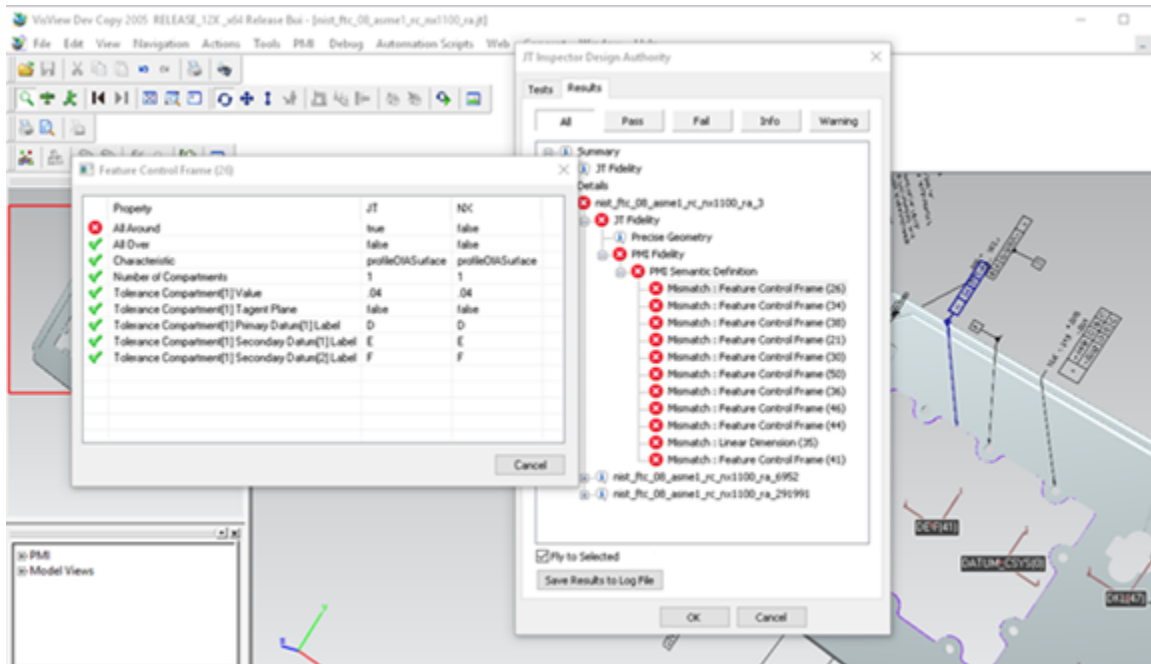


The example below shows the results when the graphical display test shows a difference. JT Inspector Design Authority, has detected, that the PMI in the JT file has been misplaced relative to the CAD system.

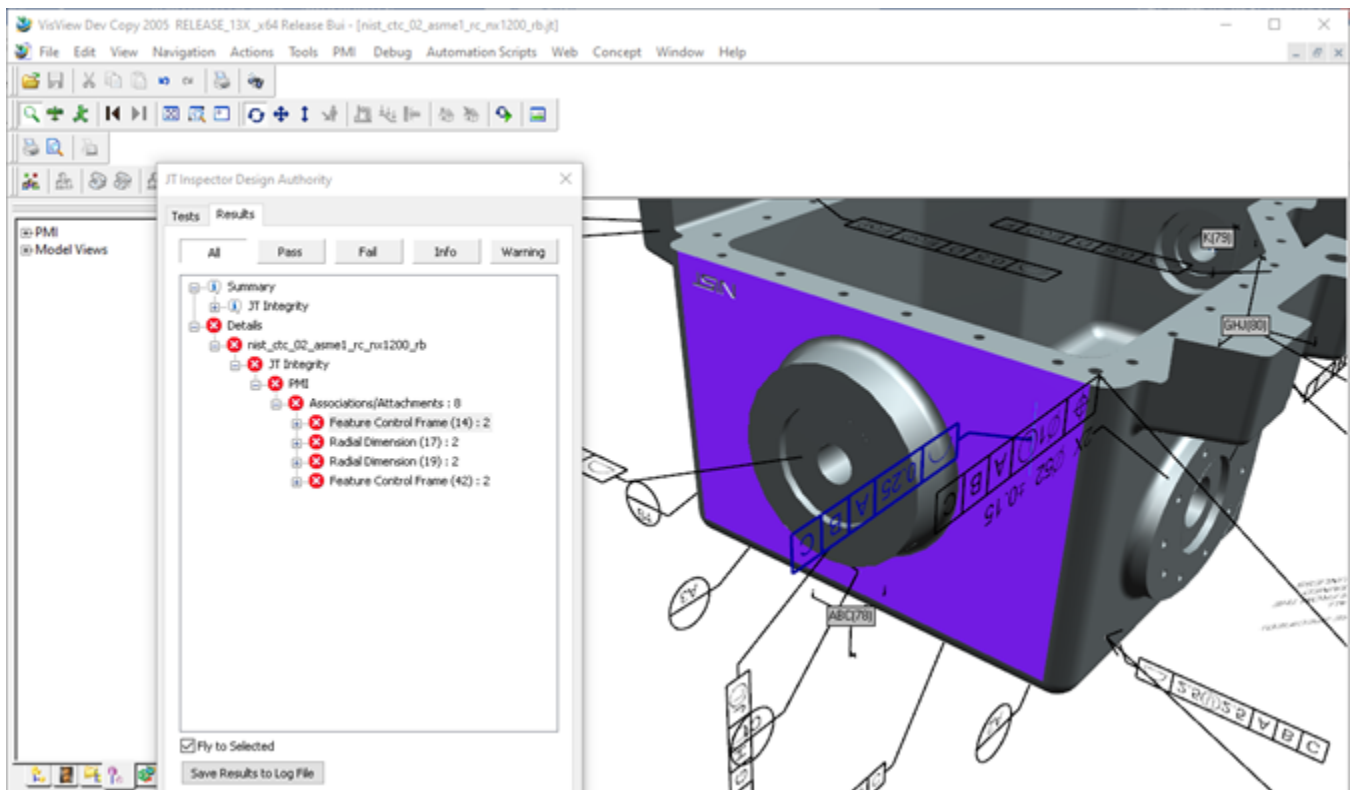
Upon selecting the failure – Mismatch: Datum Feature Symbol F(238), the pertinent PMI is highlighted (shown in blue below). Additionally, the PMI display as it exists in the CAD system is rendered in red.



The example shown below shows the results when the PMI Semantic Definition test fails. JT Inspector Design Authority, has detected, that the PMI Semantic definition is different in JT relative to the source file. Upon selecting the PMI, Feature Control Frame (26), a dialog is shown comparing the semantic definition between the JT and the source file.



The example shown below shows the results when the JT Integrity's PMI Associated/Attached Entities test fails. JT Inspector, has detected, that there are associations/attachments failures for different PMI. Upon selecting a PMI, Feature Control Frame (14) in the JT Inspector Results, the PMI in the display window is highlighted. And by checking the "Fly to Selected" box and selecting the PMI, the PMI is zoomed to its origin giving a closer look.



Save inspection results to a file

Inspection test results appear in the **Inspector** dialog box. You can also save the results to file.

1. Choose **Actions**→**Inspector**→**JT Inspector**.

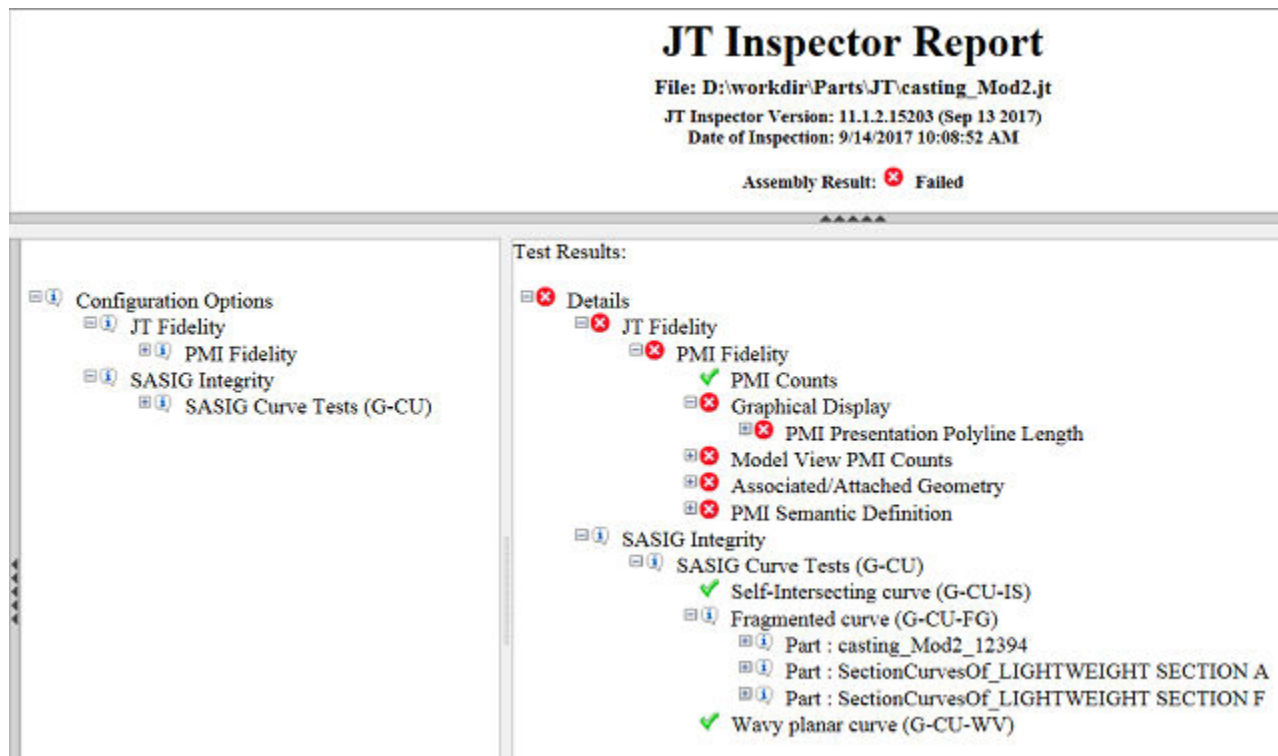
- or -

If you have the **JT Inspector Design Authority license**, choose **JT Inspector Design Authority**.

2. In the **Inspector** dialog box, on the **Results** tab, select **Save results to log file**.
3. Click **Browse** to open a file browser dialog box and select a filename and path for the file.

Note:

- By default, test results only appear on the **Results** tab of the **Inspector** dialog box.
- You can also save an Encoded XML report that includes all the results shown in the inspector dialog – summary as well as details.
- Alternatively, you can save the results to a formatted HTML report. HTML report contains less detail than the XML report and is meant to serve as a quality record.



Check geometry fidelity of translated JT files

Overview of validating JT fidelity

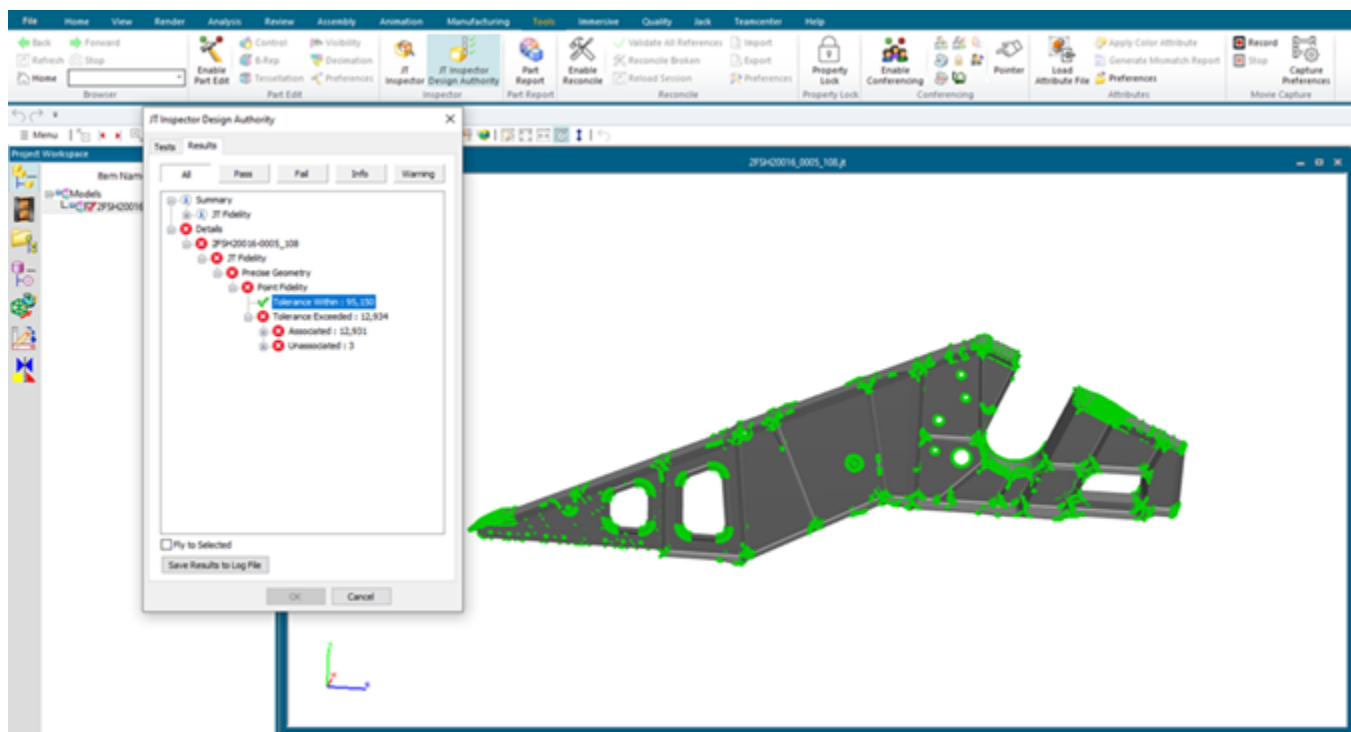
Note:

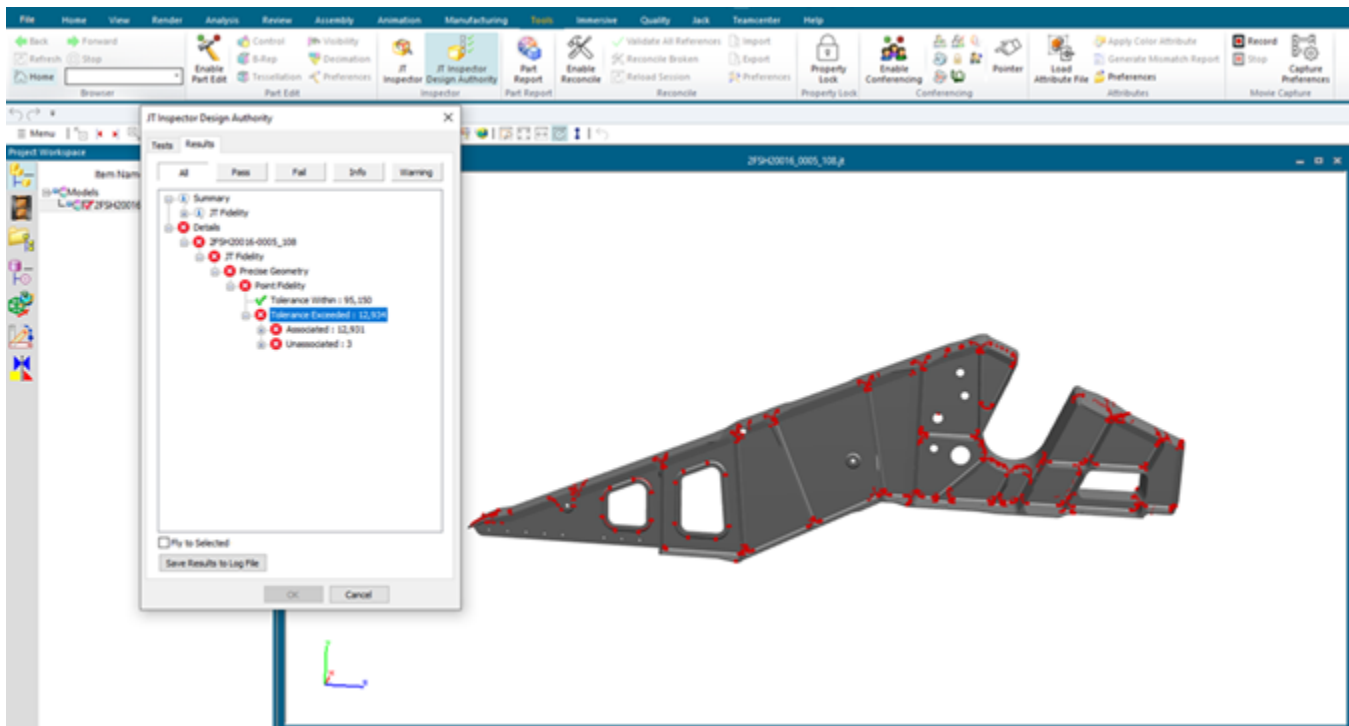
Checking Cloud of Points fidelity requires JT Inspector Design Authority license (jt inspect workflow) and the feature is supported in all standalone viewers.

Using the JT Inspector, you can validate the geometric fidelity of a translated jt file by comparing its surfaces to reference points in a point cloud file. The JT Inspector computes the distance from corresponding faces, ensuring that these distances are within a user-specified tolerance.

The point cloud file is generated during the translation of the original CAD file. The JT Inspector can read point cloud data from a PLM XML file, from the validation data within a STEP or STL file, or infer points from the tessellation vertices of a second JT file.

Selecting the Point Fidelity tests in the Inspector tree highlights the point cloud in the Viewing window. Points that pass tolerance are displayed in green. Points that exceed tolerance are displayed in red.





Point Fidelity tests are very helpful when creating scripts for translating data. It can help you view how different settings impact the results of the jt files. You can also run point fidelity tests on data launched from Teamcenter.

Specify Cloud of Points Fidelity test parameters

Note:

You must have the JT Inspector Design Authority license to run the Cloud of Points Fidelity test.

Before running a Cloud of Points Fidelity test, load the point cloud data file against which to compare the .jt file and set the tolerance.

The point cloud data can be in PLM XML, JT, STL, or STP format.

1. Choose **Actions**→**Inspector**→**JT Inspector Design Authority**.
2. From the list of fidelity tests, select **Cloud Of Points**.
3. Click **Preferences** and the **Cloud of Point Preferences** show up at the bottom of the dialog box.
4. In the **Point Cloud** dialog box, click **Browse** to navigate to a point cloud file and click **Open**.
5. In the **Tolerances** box, set a tolerance value, which is the distance, in modeling units, allowed from a perfect point in the point cloud.

- Click **OK**.

Run the Cloud of Points Fidelity test

Note:

You must have the JT Inspector Design Authority license to run the Cloud of Points Fidelity test.

- Choose **Actions**→**Inspector**→**JT Inspector Design Authority**.
- If necessary, **set** the Point Fidelity test preferences.
- In the **Inspector** dialog box, on the **Tests** tab, **specify the parts or assemblies** to be inspected.

Note:

A point cloud data file can be created and associated with each part in an assembly, or it can be one group of points associated with the entire assembly. Only PLM XML or JT can provide point clouds for the entire assembly, while STL and STP can only provide point clouds for a single part.

If you have a point cloud data file that is associated with the entire assembly, select **All** or **Selected Parts and Assembly**. Otherwise, if you leave the default, the JT Inspector returns no results.

- On the **Tests** tab, select **Cloud of Points**.
- Click **OK**.

Review the Point Fidelity test results

- Run** a Point Fidelity test.
- In the **Inspector** dialog box, in the **Results** tab, expand or collapse the **Summary** and **Details** for **Point Fidelity** by selecting the plus or minus symbols.
- Do one of the following:

To	Do this
Display the points within tolerance	In the Results tab, expand Details to Point Fidelity and select the Tolerances Within .

To	Do this
	The points within tolerance are displayed in green.
Display the points that deviate from the original model	In the Results tab, expand Details to Point Fidelity and select the Tolerances Within Exceeded. The point that deviate from the original CAD model are shown in red.

4. Performing Clearance analysis

What is Clearance?

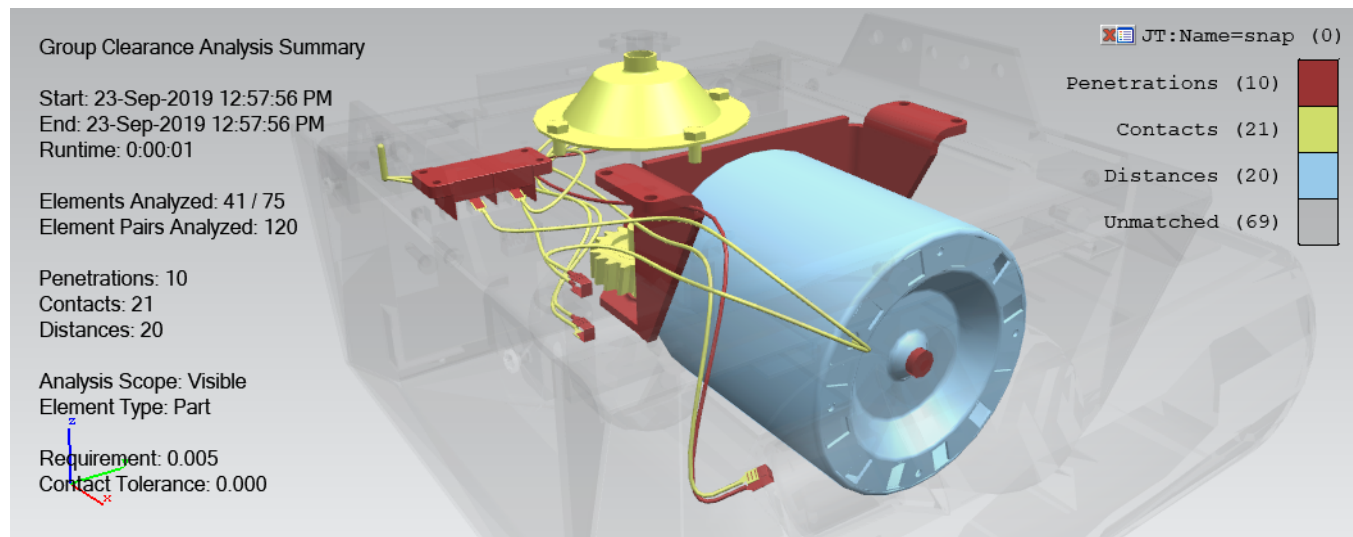
Clearance analysis

Clearance analysis identifies points of contact and/or penetration between elements or groups in your 3D model. Elements may be parts, leaf components, or end items.

The clearance requirement is the minimum allowed distance between any two elements within the analysis scope. Clearance requirements are measured in the same units as your 3D model. Elements that are in contact have a clearance requirement of 0.

Clearance analysis involves:

- Specifying the clearance settings that affect the type, precision, and nature of the analysis.
- Performing the clearance analysis on your model.
- Analyzing clearance violations (results) in the Viewing window.
- Disposition clearance issues by assigning a priority, status, and owner to element pairs on the **Results** list.



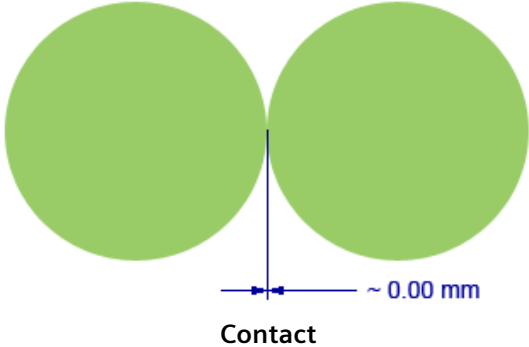
Clearance results types

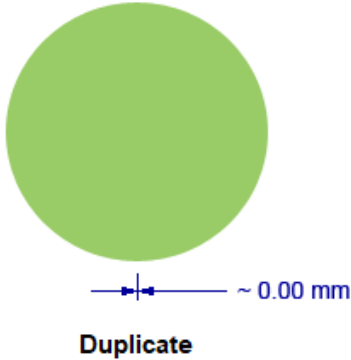
When you perform a clearance check, any two parts or elements that are closer to one another than a specified distance are identified as an issue for further investigation. The **Results** list displays the

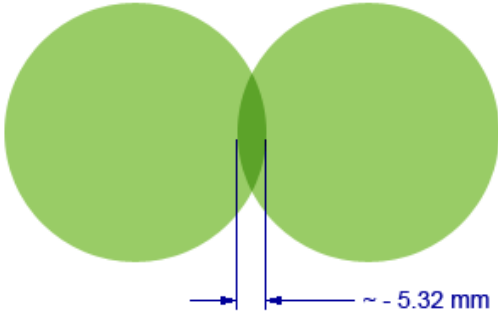
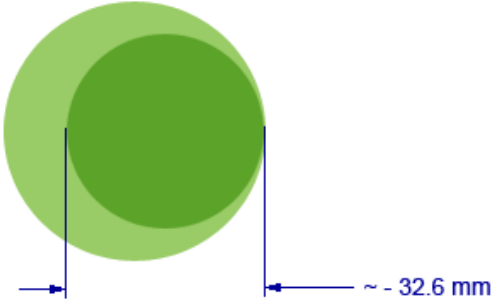
clearance issues identified by the analysis, along with the distance value between element pairs and other relevant information such as the interference type.

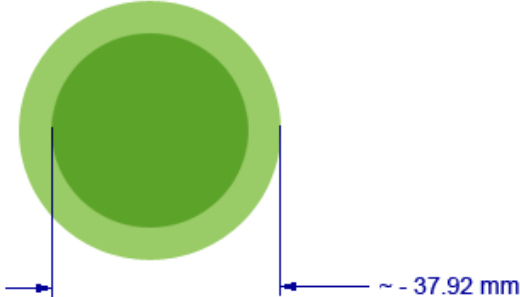
Clearance interferences can be *soft* or *hard*. Soft interferences are distance violations. They occur when the distance between two elements is equal to or closer than the specified minimum distance. Elements involved in a soft interference do not touch or penetrate one another. Hard interferences are contact or penetration violations. They occur when two elements touch or occupy the same geometric space.

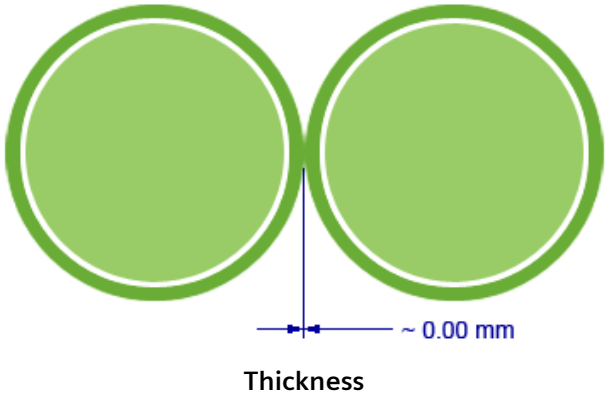
The table below describes each clearance type and provides examples to help you understand the criteria used to identify them.

This type	Appears in the results list as	Indicating this
Contact	C	A hard interference. The elements are in contact but do not penetrate. The distance result is always ~ 0.0. Note: Clearance results are approximate if your elements do not consist of precise geometry. Clearance results of geometrically similar parts that overlap may not recognize the contact if the individual polygons in the tessellated versions of the parts exactly correspond to each other.  Contact
Distance	D	A soft interference. The elements are closer than the specified minimum clearance requirement, but do not come into contact with or penetrate one another. The distance result is always a value greater than ~ 0.0.

This type	Appears in the results list as	Indicating this
		 Distance
Duplicate	Dup	The clearance calculation has determined that the elements are duplicates based on the bounding box, surface area, volume, and center of gravity. If these four attributes are the same then the parts are considered duplicates.  Duplicate
Failure	F	The clearance calculation has failed to complete. This clearance type can appear only in results generated in Batch Mode, most likely with ClearanceDB analysis. This may happen if the Clearance Calculator crashes.
Penetration	P	A hard interference. One element penetrates or is completely contained by another. The distance result is always a value less than ~ 0.0. Note: Penetration values are always approximate, even when using precise geometry. Note: Penetrations are not detected automatically unless the Distinguish between Contact and Penetrations check

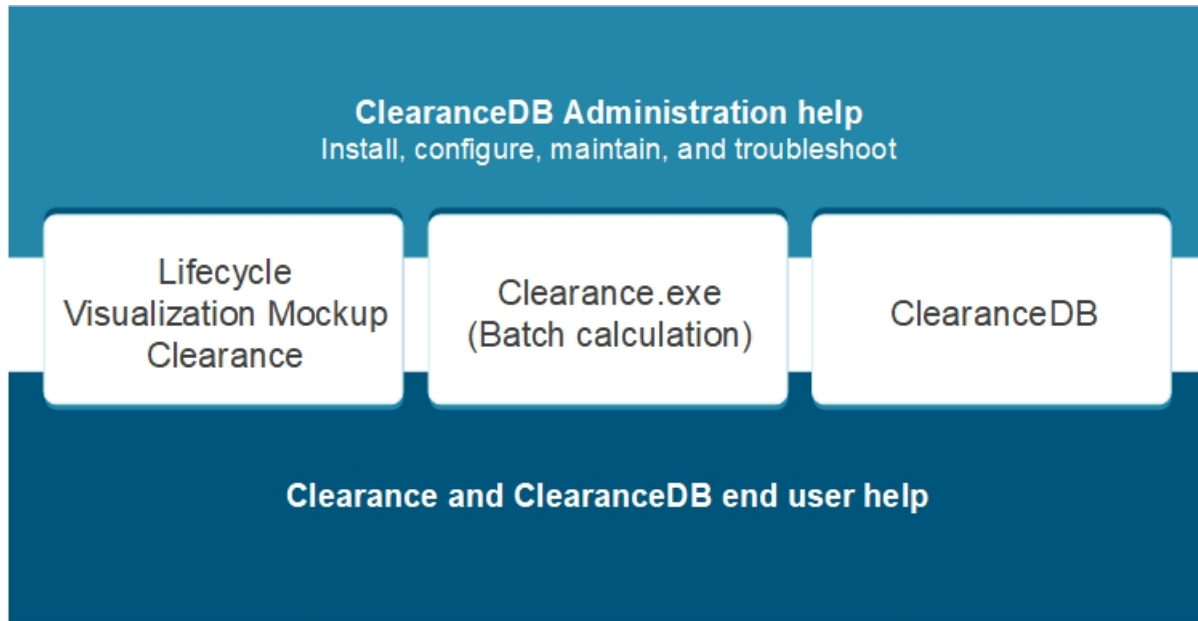
This type	Appears in the results list as	Indicating this
		box is selected in the calculator settings. If this option is not turned on, penetrations are categorized as the Unknown clearance type, and you must perform further analysis to identify the issue as a penetration.  Partial penetration For a clearance issue that consists of a pair of elements where one element is completely contained within another, the distance value is the approximate penetration depth of the contained element. If the minimum distance between elements is ~ 0.0 (the elements are in contact with one another), and the elements contain NURBS data, the penetration depth is the length of the bounding box diagonal of the inner element. Note: If the minimum distance between elements is ~ 0.0 and the parts do not contain NURBS data, the issue is classified as a contact.  Full penetration (elements in contact) If the minimum distance between elements is greater than ~ 0.0 (the elements are not in contact with one another), the

This type	Appears in the results list as	Indicating this
		penetration depth is the minimum distance plus the distance the minimum distance vector extends to the further side of the inner elements' bounding box.  Full penetration (elements are apart) Note: • Completely contained elements are categorized as penetrations only if the outer element is geometrically closed. • The inner element must consist of tessellated geometry. An element pair with an inner element that consists only of wireframe geometry is not identified as a penetration. • An inner element with a material thickness value that causes the element to extend beyond the outer element is considered completely contained and identified as a penetration.
Thickness	T	A violation where material thickness is a factor of the calculation. Further analysis is required to detect whether an issue involving material thickness is a soft or hard interference. The distance result can be a value less than or greater than ~ 0.0. Note: Element pairs with material thickness values that come into contact or penetrate one another are identified as the Unknown clearance type. Contact, penetration, and distance line analysis are not available for clearance

This type	Appears in the results list as	Indicating this
		issues that contain element pairs with material thickness values. 
Unknown	U	A hard interference. Further analysis is required to classify as a contact or penetration.

What is ClearanceDB?

Teamcenter Clearance Database (ClearanceDB) enables customers to detect, manage, and resolve spatial interference issues as they arise during a product's lifecycle. By automating this process and constantly re-evaluating on a regular basis, customers can identify design flaws that may have otherwise delayed the release of new product offerings, increased costs, or jeopardized quality.



Capabilities

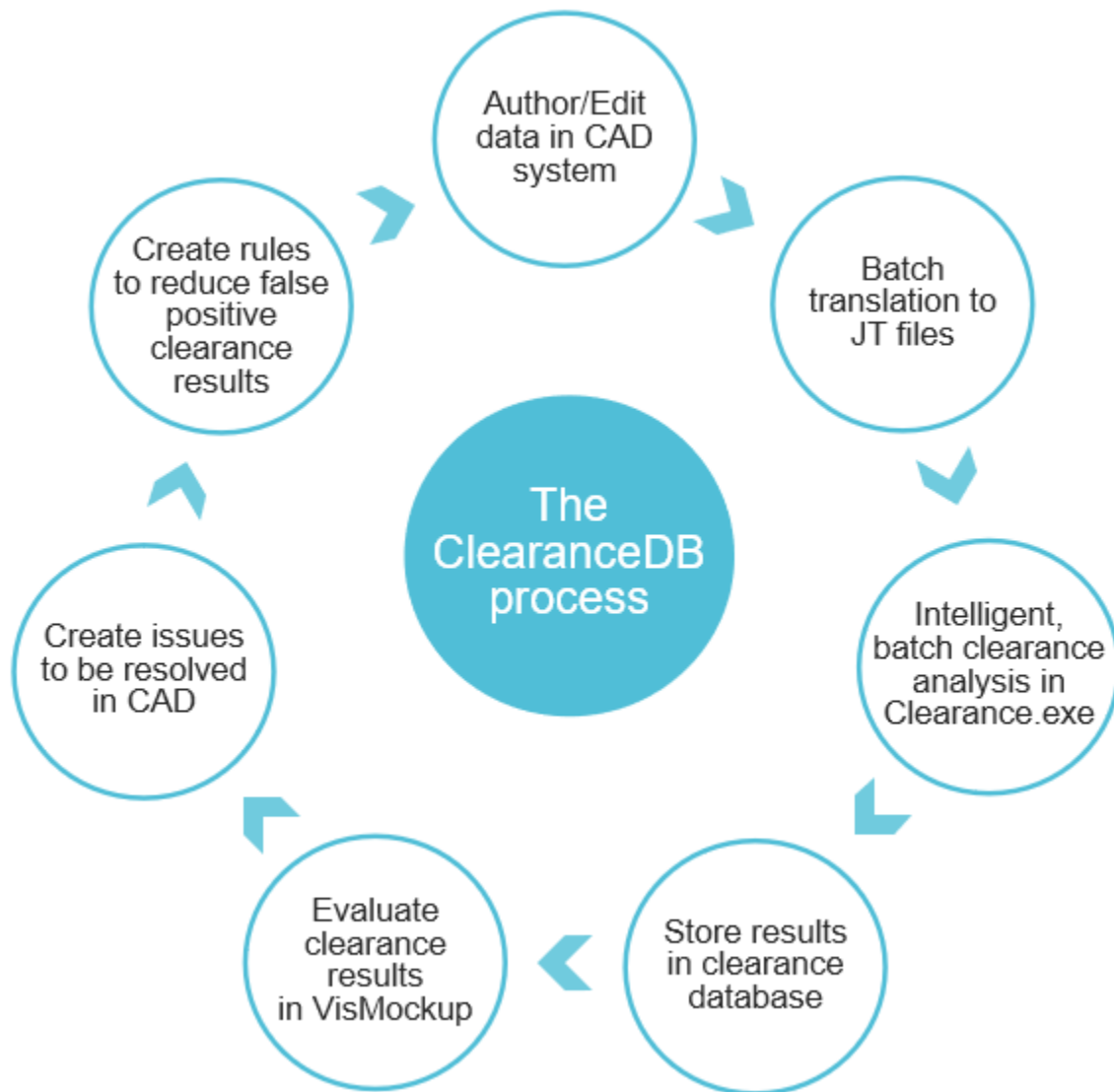
The Clearance db server option provides the following capabilities:

- Completely automated, intelligent batch clearance analysis that is continuously updated.
- Analysis results are stored in the Clearance database.
- The Clearance database can be accessed by users at any time to understand the number and severity of clearance issues for any product, or to investigate specific issues in Teamcenter lifecycle visualization mockup.
- Customers can analyze and manage clearance issues across multiple product configurations, derivatives, and options by integrating the clearance analysis process into a Teamcenter-managed product lifecycle.

Benefits

- Automates the checking and reporting process to facilitate faster, repeatable, and more accurate clearance evaluations.
- Reduces development cost by enabling engineers to catch design flaws early in the lifecycle.
- Fosters innovation by enabling designers and engineers to focus on products instead of DMU (Digital Mockup) issues.
- Reduces design cost by enabling design engineers to digitally integrate the entire product.

The ClearanceDB process



Configuring the clearance analysis

Introduction to configuring clearance analysis

To perform clearance analysis, you must first specify some basic clearance settings, including the type of element to analyze, the minimum clearance requirement, and the clearance calculator settings.

Overview of the clearance calculator settings

Calculator settings specify the precision and nature of the clearance check. You can choose to

- Specify the **analysis granularity**. Choose from elements (parts, leaf components, and end items) and CAD components.
- Use **NURBS data** (if it is available in the model) or tessellated data to calculate points of contact.

Tip:

To determine whether your model includes NURBS data, right-click a part. If the options **Show Vertices** and **Show Edges** are available, the model includes NURBS data; if the options are gray, the model does not include NURBS data.

- For Results Granularity on Selected and Dynamic clearance, specify whether **violations are reported** at the selected assembly level or at a lower level within the assembly, such as at the part level.
- Automatically analyze each clearance issue to identify **points of contact or penetration**.

Tip:

If you choose to automatically analyze each clearance issue, you can also specify a tolerance for penetrations, so that any penetration value that is less than the tolerance is classified as a contact.

- Apply **material thickness** to sheet parts for analysis.
- Control how clearance calculations are performed for **4G data with multiple subsets**.

Set the clearance requirement

Element pairs that are closer than the minimum clearance requirement will be identified during the clearance analysis.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Set Clearance** .
3. In the **Set Clearance** dialog box, enter a value for the minimum allowed distance between elements.

The distance unit is determined by the current 3D document and cannot be modified in Lifecycle Visualization.

Tip:

A clearance requirement value of 0 identifies only elements that are in contact with or penetrating other elements. In order to create distance (D) violations, a requirement greater than 0 must be entered.

4. To set the value so that it is in effect for the open model and subsequent open models, select **Set as Default**.

If this option is *not* selected, the value you enter applies only to this model while it is open. When you re-open this model or open subsequent models, the current default value is used. (Upon installation, the default value is 0.)

5. Click **OK**.

The clearance requirement is set. When you perform a clearance check, only those elements within the clearance requirement are identified.

Set the clearance scope

On the **3D Clearance** toolbar, specify the scope of the clearance check.

Use this option	To do this
All Elements 	Identify and load all elements in your model that are within the set clearance requirement, even if the elements are not loaded.
Loaded Elements 	Identify all loaded elements that are within the set clearance requirement, even if the elements are not visible.
Visible Elements 	Identify all visible elements that are within the set clearance requirement.

Set the visibility action for clearance analyses in the 3D view

Control how violating elements are displayed in the Viewing window.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. Choose **Clearance→Visibility Action**, and then choose one of these options:

Use this option	For this
No Action	Do not change the display of the Viewing window.

Show elements	Show only the violating elements in the Viewing window.
Hide elements	Hide the violating elements in the Viewing window.
Select elements	Select the violating elements in the Viewing window.
Visual Report	In the Viewing window, display a summary of the violating elements in a visual report. Note: Optionally, to view a breakdown of where the violating elements occur within the product structure, select the Display the Clearance Results and Clearance Graph columns in the assembly tree when Clearance is enabled checkbox on the Clearance Results Preferences dialog box. The columns display in the Assembly view during clearance analysis. These columns may be viewed independently from the visual report. Use the Clearance Results Preferences dialog (Result page) to define whether the visual report displays the number of violating <i>element pairs</i> (Results option) or the number of violating <i>elements</i> (Elements option).

3. Perform a macro (matrix, selected, or group) clearance analysis.

In the Viewing window, the violating elements are displayed as you specified.

Clearance calculator settings

Control the analysis granularity

You can control the granularity of clearance analysis. Choose from elements (parts, leaf components, and/or end items) and CAD components.

Note:

If the analysis granularity is set to leaf component or end item, all of the parts below those nodes are treated as a single, monolithic element.

1. Choose **Clearance→Preferences→Calculator Settings**.
2. In the **Analysis Granularity** section of the **Clearance Calculator Settings** dialog box, choose from the following:

Use this option	To do this
Parts (Always On)	Perform analysis on all parts in the model. Note: You cannot disable parts as clearance elements, because parts are the lowest-level items (atoms) in the product structure. Note: To define the granularity of the analysis results, select either Assemblies or Parts within the assemblies under Results Granularity.
Leaf Component	Perform analysis on leaf components in the model. Note: To define the granularity of the analysis results, select either Elements or Parts within the elements under Results Granularity.
End Item	Perform analysis on end items in the model. Note: To define the granularity of the analysis results, select either Elements or Parts within the elements under Results Granularity.
CAD Component	Perform analysis on CAD components in the model.

- Click **OK**.

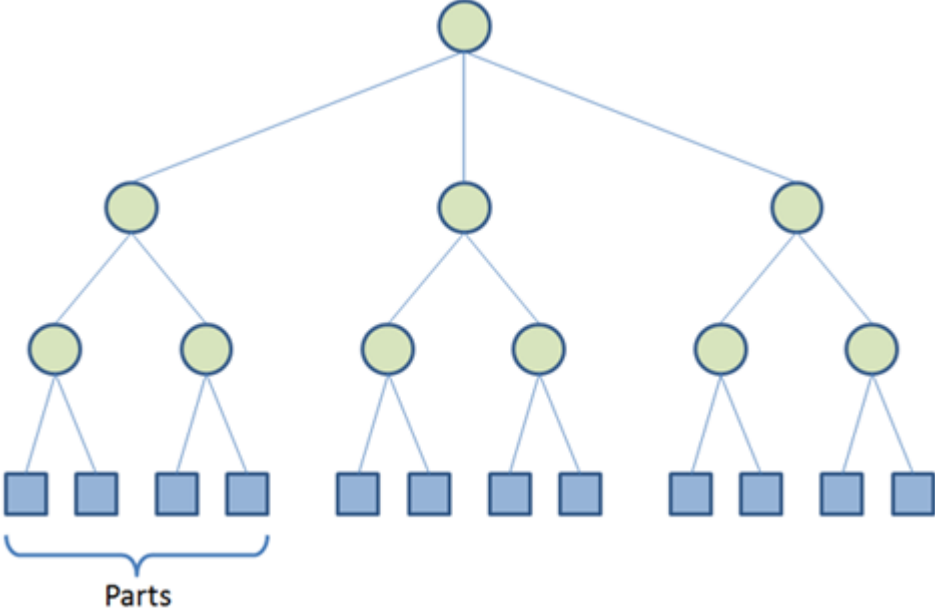
Clearance elements and CAD components

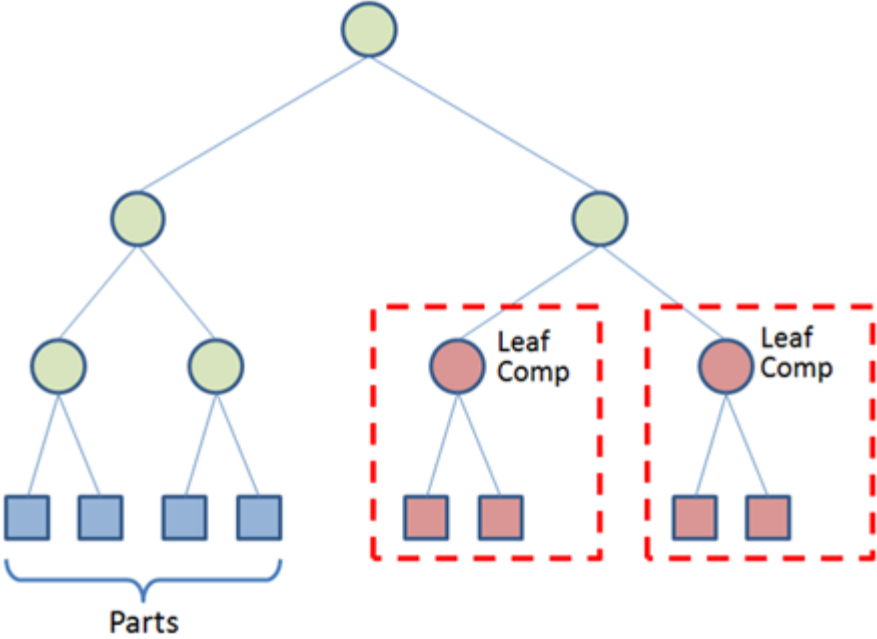
Elements and CAD components are the parts and branches of product structure recognized for clearance analysis in Teamcenter lifecycle visualization. They are used to control the granularity of the analysis.

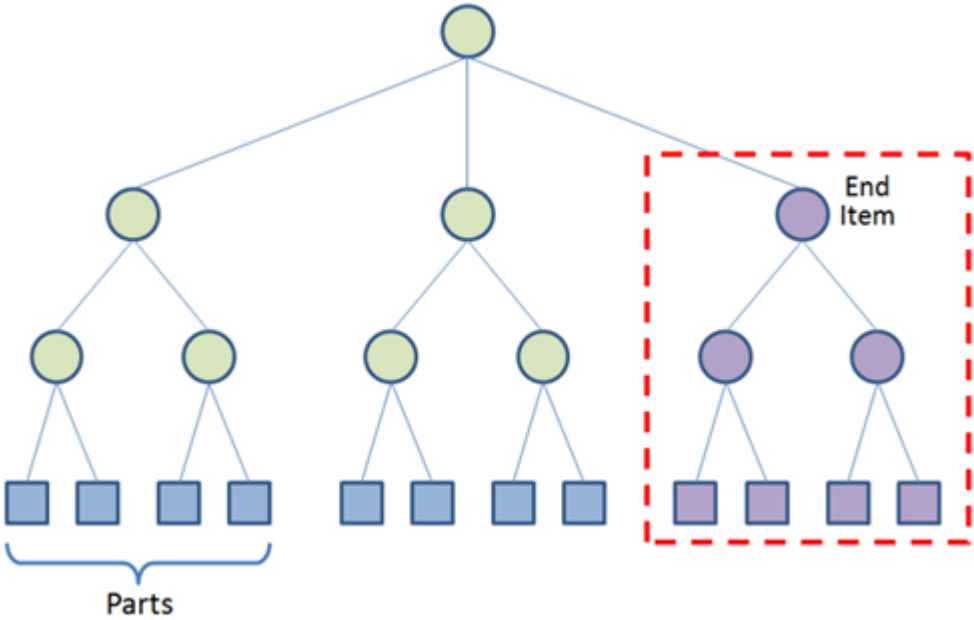
There are three element types: parts, leaf components, and end items. Any combination of these elements can exist in the product structure for use during a clearance analysis. However, the part element type is always turned on.

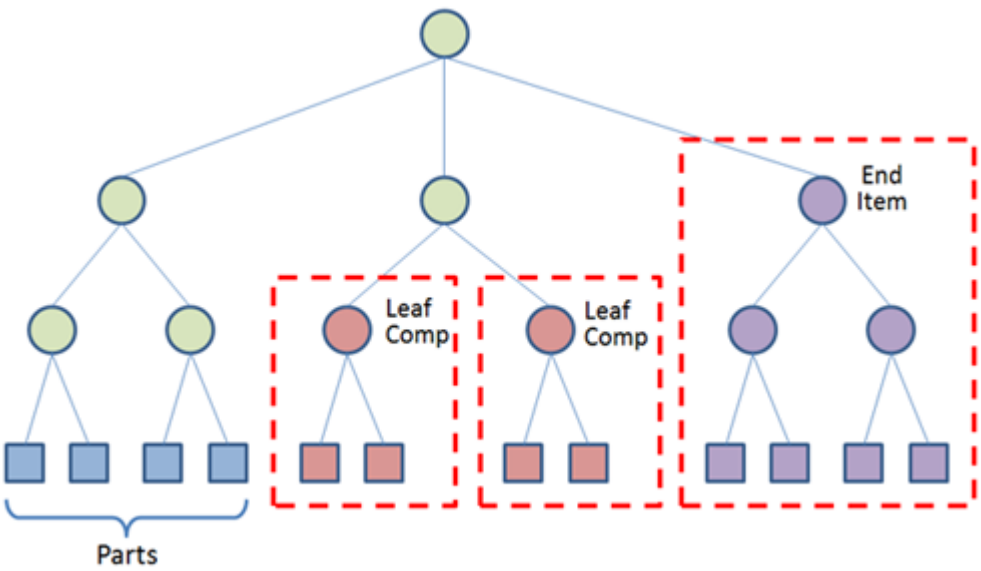
If an element contains other elements within it, no clearance issues between those elements are reported in the clearance results list. Also, any two leaf components or end items can have many clashes between them, but only the single, most severe result between any two elements is reported.

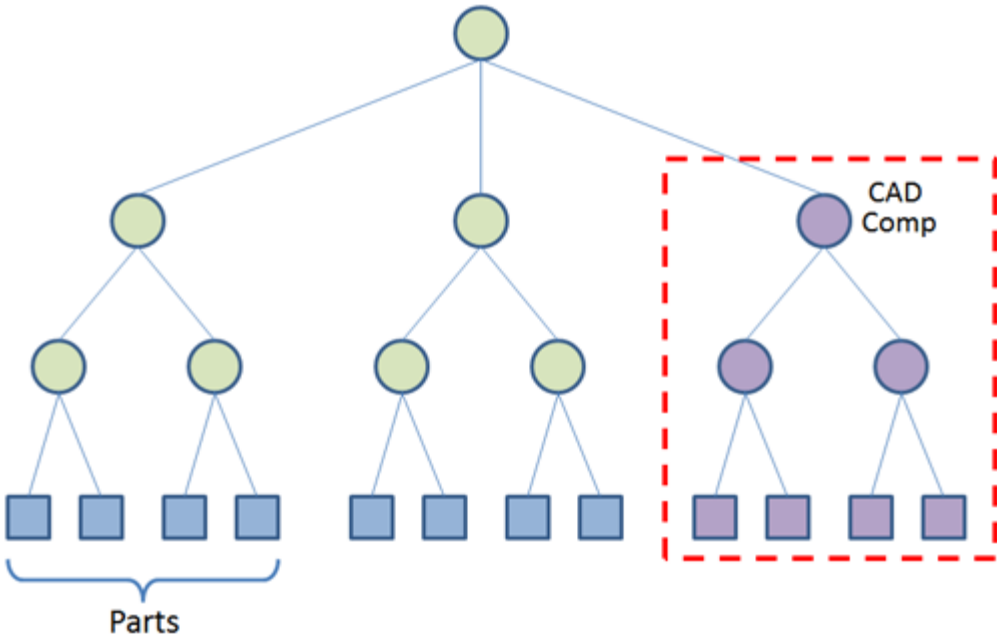
The items that appear in the clearance results list are always elements. A CAD component is essentially an end item that allows any number of clearance results between itself and other nodes. For this reason, a CAD component is not considered an element.

Element type	Behavior
Part	 Analysis The lowest level elements in the clearance analysis are parts. Analysis is always conducted between part elements, unless the parts are overridden by other element types (leaf components and end items) or CAD components. Results

Element type	Behavior
	Only issues between parts are displayed in the clearance results, even if leaf components, end items, or CAD components exist in the structure.
Leaf component	 Analysis The lowest level elements in the clearance analysis are parts, unless they are overridden by leaf components. In that case, all of the parts underneath the leaf component node are considered to be part of the same, monolithic element. Results Only the single, most severe result between each leaf component and other elements is displayed. That result appears in the clearance results list as the leaf component itself (not the parts within the leaf component).

Element type	Behavior
End item	 Analysis The lowest level elements in the clearance analysis are parts, unless they are overridden by an end-item. In that case, all of the parts underneath the end item node are considered to be part of the same, monolithic element. Results Only the single, most severe result between each end item and other elements is displayed. That result appears in the list as the end item itself (not the parts within the end-item). Note: End items are branches of product structure where the root node contains the metadata <code>__PLM_END_ITEM=True</code>.

Element type	Behavior
Leaf components and end items	 Analysis The lowest level elements in the clearance analysis are parts, unless they are overridden by a leaf component or end item. In that case, all of the parts underneath the “parent” node are considered to be part of the same, monolithic element. Results No issues are reported between elements within the leaf component or end item. Only the single, most severe result between each leaf component or end item and

Element type	Behavior
	other elements is displayed. That result appears in the list as the root item (not the parts within the root item).
CAD components	 Analysis The lowest level elements in the clearance analysis are parts, unless they are overridden by a CAD component. In that case, all of the parts underneath the CAD component node are considered to be part of the same monolithic component. Results No issues are reported between elements within the CAD component. However, any number of issues may be reported between elements within the CAD component and other elements in the model. Those results appear in the list as the elements themselves (not the CAD component). Note: CAD components are branches of product structure where the root node contains the metadata <code>__PLM_CADCOMPONENT=True</code>.

Specify the contact/penetration analysis

- 1. Choose **Clearance**→**Preferences**→**Calculator Settings**.

2. In the **Contact/Penetration Analysis** section of the **Clearance Calculator Settings** dialog box, set any of the following preferences:

Select this option	To do this
Use NURBS	Use NURBS to calculate contact/penetration points between element pairs. Note: Using NURBS to calculate the contact/penetration points provides more accurate results, but may significantly affect the calculation performance.
Distinguish between Contact and Penetrations	If contact/penetration points are found for an element pair, determine if the result type for the pair should be contact or penetration . If this option is not selected, the result type for all pairs with contact/penetration points is U (unknown). In that case, the final type for U element pairs is determined when each pair is analyzed.
Contact Tolerance	If Distinguish between Contact and Penetrations is selected, optionally enter the distance two elements can penetrate and still be classified as a contact instead of a penetration.

3. Click **OK**.

Distinguishing between contacts and penetrations

By default, the **Distinguish between Contact and Penetrations** option is turned on, and when you run a clearance check all issues are automatically classified as contacts or penetrations.

Type	Element Number1	Element Number2	Result
C	49-03-5502_Magnet Retainer	49-03-5501_Magnet	~ 0.0000
P	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ -0.0000
C	49-03-5502_Magnet Retainer	49-03-5502_Magnet Retainer	~ 0.0000
C	49-03-5501_Magnet	49-03-5502_Magnet Retainer	~ 0.0000
P	49-03-5502_Magnet Retainer	49-03-5201_Brush Holder	~ -0.0042
P	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ -0.0000
P	49-03-5502_Magnet Retainer	49-03-5201_Brush Holder	~ -0.0042
P	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ -0.0000
P	49-03-5306_Base Plate	49-04-5005_Gear1	~ -0.0040

Distinguish between Contact and Penetrations turned on

If you turn off the **Distinguish between Contact and Penetrations** option, clearance issues are designated as the **Unknown** clearance type, and you must right-click the issue in the **Results** list and choose **Analyze Clearance** for more detailed information.

Type	Element Number1	Element Number2	Result
U	49-03-5502_Magnet Retainer	49-03-5501_Magnet	~ 0.0000
U	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ 0.0000
U	49-03-5502_Magnet Retainer	49-03-5502_Magnet Retainer	~ 0.0000
U	49-03-5501_Magnet	49-03-5502_Magnet Retainer	~ 0.0000
U	49-03-5502_Magnet Retainer	49-03-5201_Brush Holder	~ 0.0000
U	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ 0.0000
U	49-03-5502_Magnet Retainer	49-03-5201_Brush Holder	~ 0.0000
U	49-03-5301_Rotor Shaft	49-03-5502_Magnet Retainer	~ 0.0000
U	49-03-5306_Base Plate	49-04-5005_Gear1	~ 0.0000

Distinguish between Contact and Penetrations turned off

For greater control over analysis configured to use the **Distinguish between Contact and Penetrations** option, you can specify a **Contact Tolerance** value. The **Contact Tolerance** option enables you to prevent issues with depths less than a specified amount from being classified as penetrations.

For example, you perform a clearance check that reveals a large number of penetrations with depths less than .01 mm, the identification of which you consider insignificant to your product design. To prevent these issues from being classified as penetrations, you specify a **Contact Tolerance** value of .01. Now when you run your clearance check again all issues with penetration depths less than .01 mm are classified as contacts.

Type	Element Number1	Element Number2	Result
P	49-03-5601_Brush Reel	49-03-5101_Motor Housing	~-0.0014
P	49-03-5101_Motor Housing	49-03-5201_Brush Holder	~-0.0105
P	49-03-5306_Base Plate	49-03-5101_Motor Housing	~-0.0006
P	49-03-5101_Motor Housing	49-05-5104_Trigger Button	~-0.0010
P	49-03-5405_Bearing Wrap	49-03-5101_Motor Housing	~-0.0004
P	49-03-5404_Bearing Top	49-03-5101_Motor Housing	~-0.0010
P	49-03-5101_Motor Housing	49-03-5202_Brush Cap Screw	~-0.0135
P	49-03-5101_Motor Housing	49-03-5202_Brush Cap Screw	~-0.0065
P	49-03-5101_Motor Housing	49-03-5004_Armature Shield	~-0.0003

Contact Tolerance value of 0

Type	Element Number1	Element Number2	Result
C	49-03-5601_Brush Reel	49-03-5101_Motor Housing	~ 0.0000
P	49-03-5101_Motor Housing	49-03-5201_Brush Holder	~-0.0105
C	49-03-5306_Base Plate	49-03-5101_Motor Housing	~ 0.0000
C	49-03-5101_Motor Housing	49-05-5104_Trigger Button	~ 0.0000
C	49-03-5405_Bearing Wrap	49-03-5101_Motor Housing	~ 0.0000
C	49-03-5404_Bearing Top	49-03-5101_Motor Housing	~ 0.0000
P	49-03-5101_Motor Housing	49-03-5202_Brush Cap Screw	~-0.0135
C	49-03-5101_Motor Housing	49-03-5202_Brush Cap Screw	~ 0.0000
C	49-03-5101_Motor Housing	49-03-5004_Armature Shield	~ 0.0000

Contact Tolerance value of .01

Enable intersection volume calculations

1. Choose **Clearance**→**Preferences**→**Calculator Settings**.
2. In the **Intersection Volume** section of the **Clearance Calculator Settings** dialog box, set the following preference:

Select this option	To do this
Calculate	Calculate the intersection volume for all penetrating element pairs (type P), in the current document units. The intersection volume for all other result types is zero. Note: • Calculating the intersection volume may significantly affect the calculation performance. • This option requires that XT B-Rep data is included in your JT files. If an XT B-Rep is not available, the volume cannot be calculated.

3. Click **OK**.

Specify violation reporting level

Use this option to specify the level at which violations are reported when using Selected or Dynamic clearance analysis.

Example:

Selected clearance report for the assembly handle_assembly_1.

Item Name	Typ	Element Number1	Element Number2
Clearance Results (9/9)			
Result (1/9)	C	handle_assembly_1	body
Result (2/9)	D	handle_assembly_1	rotor
Result (3/9)	D	handle_assembly_1	Part1
Result (4/9)	D	handle_assembly_1	spool
Result (5/9)	D	handle_assembly_1	drag_knob
Result (6/9)	D	handle_assembly_1	bail_wire2
Result (7/9)	D	handle_assembly_1	Part5
Result (8/9)	D	handle_assembly_1	Part4
Result (9/9)	D	handle_assembly_1	button

Violation results when **Report violations at the selected assembly level only** is selected.

Item Name	Typ	Element Number1	Element Number2
Clearance Results (27/2...			
Result (1/27)	C	handle1_2	body
Result (2/27)	D	handle	body
Result (3/27)	D	handle1_2	rotor
Result (4/27)	D	handle	rotor
Result (5/27)	D	handle1_2	Part1
Result (6/27)	D	handle	Part1
Result (7/27)	D	handle_2	body
Result (8/27)	D	handle1_2	spool
Result (9/27)	D	handle1_2	drag_knob
Result (10/27)	D	handle	drag_knob
Result (11/27)	D	handle	spool
Result (12/27)	D	handle1_2	bail_wire2
Result (13/27)	D	handle	Part5
Result (14/27)	D	handle1_2	Part5
Result (15/27)	D	handle_2	Part1
Result (16/27)	D	handle_2	rotor
Result (17/27)	D	handle	bail_wire2
Result (18/27)	D	handle	Part4
Result (19/27)	D	handle1_2	Part4

Violation results when **Report violations with descendants of selected assemblies** is selected.

1. Choose **Clearance→Preferences→Calculator Settings**.
2. In the **Results Granularity** section of the **Clearance Calculator Settings** dialog box, choose from the following:

Use this option	To do this
Assemblies	Reports violations on the selected assembly level, only.
Parts within the assemblies	Reports violations on any descendants (child parts) within the selected assemblies.
Elements	Reports violations on selected elements only for end items and leaf components.

	Note: The model must have end items or leaf components for this option to apply. In addition, the Leaf Component or End Item option under Analysis Granularity must also be selected.
Parts within the elements	Reports violations on any descendants (child parts) within selected end items and leaf components. Note: The model must have end items or leaf components for this option to apply. In addition, the Leaf Component or End Item option under Analysis Granularity must also be selected.

3. Click **OK**.

Specify the calculation for 4G data with subsets

You can control how clearance calculations are performed for 4G data with multiple subsets.

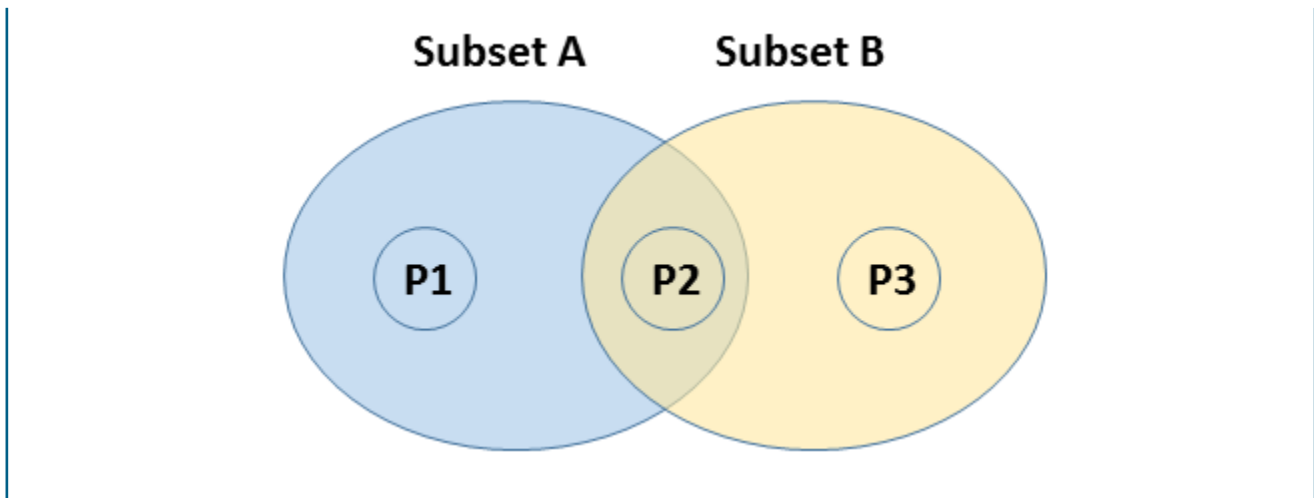
1. Choose **Clearance**→**Preferences**→**Calculator Settings**.
2. In the **Subsets** section of the **Clearance Calculator Settings** dialog box, select one of the following options to control how 4G data with subsets is calculated:
 - **Only allow issues within each subset**
 - **Only allow issues between subsets**
 - **Allow all issues regardless of subset**
3. Click **OK**.

Example:

Issues within each subset: (P1, P2), (P2,P3)

Issues between subsets: (P1, P3)

All issues: (All)

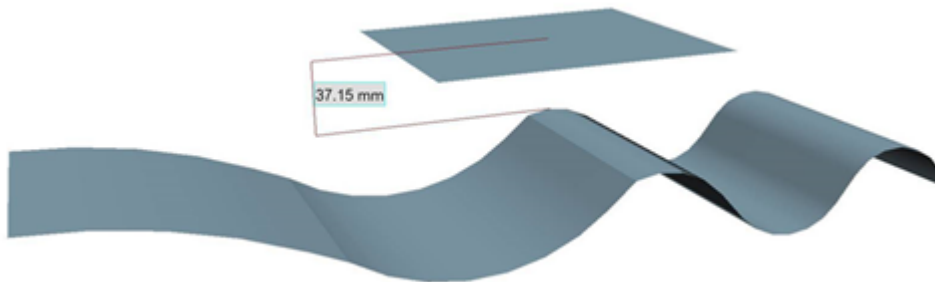


Specify the material thickness

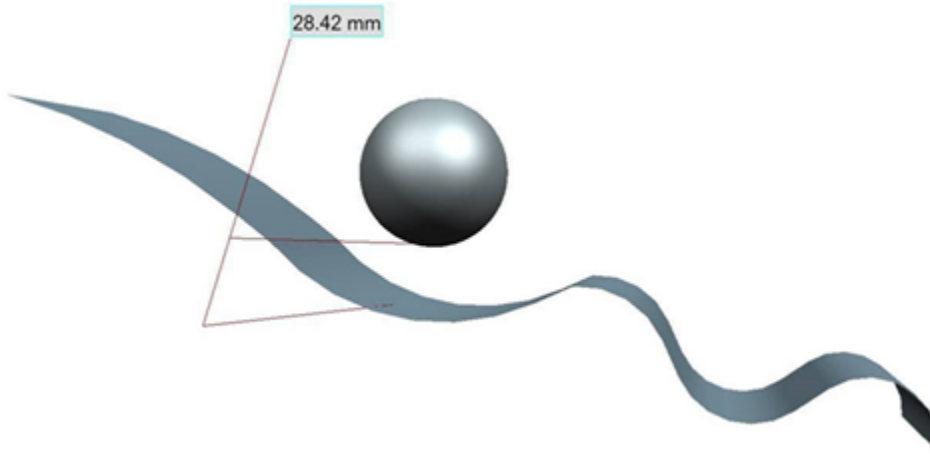
Use the clearance calculator settings to apply material thickness to sheet parts for analysis. The material thickness value is taken from part attribute metadata or from a default value that you specify. The thickness is applied to the normal side of each surface and used during the clearance analysis. Parts that are in violation based on applied thickness appear in the **Results list** with a result **Type** value of **T** (Thickness).

There are two types of thickness violations:

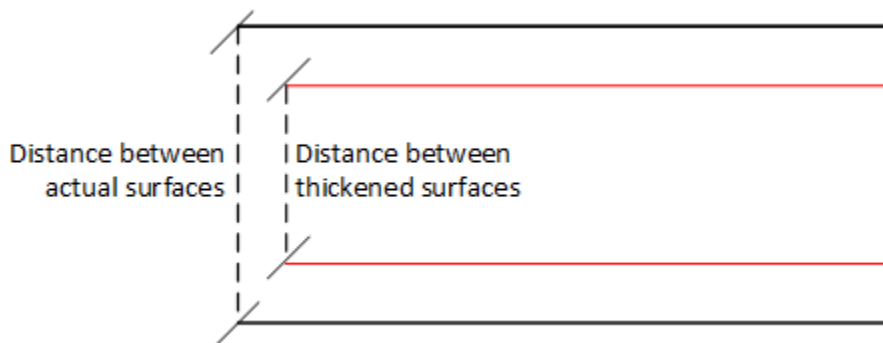
- Surface-to-surface



- Surface-to-solid



The clearance result is the distance between the actual sheet part surfaces and not the thickened surfaces.



1. Choose **Clearance**→**Preferences**→**Calculator Settings**.
2. In the **Clearance Calculator Settings** dialog box, select the **Apply Material Thickness (sheet parts only)** check box.
3. In the **Material Thickness** section, set any of the following preferences:

Use this option	To do this
(Optional) Calculate more accurate result and location (slower operation)	Set the calculation so that it does not stop at the first violation found but continues until it finds the most optimum result and location. Though this setting finds optimum results, it is a slower process.

	Note: If this option is disabled, a type (T) is reported in the Results list. If this option is enabled, a distance value (D) or an undefined value (U) is reported in the Results list.
Offset in face normal direction	Calculate the material thickness on the same side as the surface normal. Tip: Use the Two-shaded lighting option on the Preferences dialog box to determine the face normal direction. When this option is turned off, the normal face appears black.
Thickness attribute	If you have parts that contain material thickness metadata, select the attribute that is used to convey the material thickness value.
Action	If you have parts without material thickness metadata, specify how the analysis should handle parts without material thickness values. Select one of the following options: • Use Default value: Applies the value listed in Default value to parts with missing thickness attributes. • None: If the part does not have a thickness attribute, the thickness value is set at zero and thickness is not calculated for this part. • Prompt: A dialog box appears for any parts that do not have thickness attributes. Supply a thickness value for the part.
Default value	If you selected Use Default value above, enter a value to be applied to parts with missing thickness attributes.

4. Click **OK**.

When you perform a clearance check involving sheet parts, these settings are used to calculate points of clearance and penetration.

Enable multi-threaded clearance analysis

You can specify whether to perform clearance calculations in multi-threaded mode, which improves performance by taking advantage of multiple CPUs/cores on your machine (if available).

1. Choose **Clearance**→**Preferences**→**Calculator Settings**.
2. In the **Performance** section, select the **Perform multi-threaded analysis** check box.

Change the precision of clearance calculations

Select the degree of precision to be used in the clearance analysis.

1. Choose **Clearance**→**Preferences**→**Display Precision**.
2. From the **Precision** list, select the number of decimal places.
3. Click **OK**.

Create images for clearance results

During analysis, you may want to create 2D images (.jpg or .png) of the clearance results to support your clearance process. These images can then be shared on the local file system, or uploaded to the Clearance database.

The following apply to clearance result images:

- Images containing a cross section of the violating pair reflects geometry clipped on the near the side (Clip Near) of the section, with respect to the current view.
- The date and time that the images are created is appended to the image subfolder name using the format YYYYMMDD-hhmmss (24-hour clock).

Example:

C:\Clearanceimages\fishing_reel_20230731-130945

- Images will be created the next time a clearance analysis is performed and will be placed in the folder defined in the **Image Directory** field in the **Clearance Results Preferences** dialog box. The folder name will be the name of the current 3D model with the appended date and time that the image was created.

You can choose to create a single image per result, or multiple images per result.

Create a single image per clearance result

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Preferences** .
3. In the **Clearance Results Preferences** dialog box, on the General page, select the **Generate images during clearance analysis** check box.
4. In the **Image Directory** field, click **Browse** to navigate to and select an image directory.

5. (Optional) use the **Height** and **Width** fields to change the image size.
6. Click **OK**.

Create multiple images per clearance result

You can choose to have multiple 2D images created for element pairs involved in a clearance violation. For each violation type, use the advanced image option to define the number of images taken, camera zoom factor, camera angle, and whether or not a cross section of the violating pair is created. You can define as many camera settings as you wish for each violation type and each camera definition can be unique.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Preferences** .
3. In the **Clearance Results Preferences** dialog box, on the General page, do the following.
 - a. Select the **Generate images during clearance analysis** check box.
 - b. Select the **Create multiple images per result** check box.
 - c. Click **Define Images**.
4. In the **Define Clearance Images** dialog box, do the following.
 - a. Select the page for the desired clearance violation type (Penetrations, Contacts, Distances, or Duplicates) and define the camera settings by doing one of the following.

To	Do this
Add a camera definition.	Click Add . A new camera definition row is added to the table. Edit the definition to customize the camera settings.
Edit a camera definition.	A. Select the desired camera row from the table. B. Click Edit. C. In the Edit Image Settings dialog box, use the dropdown lists to define the camera Zoom %, Camera angle, and Section values for this camera definition. D. (Optional) Click the right or left arrows to navigate to the next camera definition or to the previous camera definition.

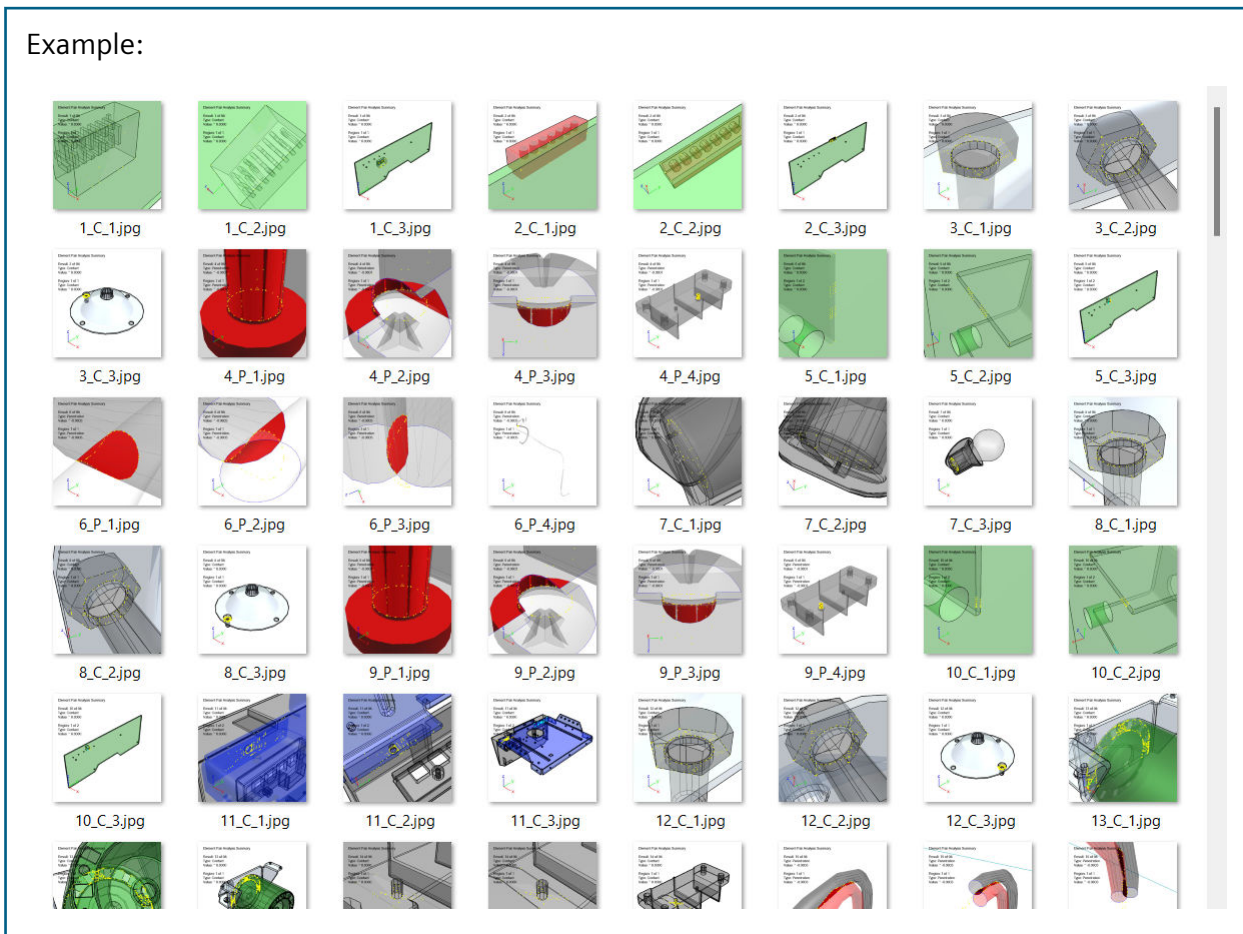
To	Do this
	E. Click OK .
Delete a camera definition.	A. Select the desired camera row from the table. B. Click Delete .

b. In the **Define Clearance Images** dialog box, click **OK**.

5. In the **Image Directory** field, click **Browse** to navigate to and select an image directory.

Images created during clearance analysis will be placed in the selected directory.

Example:



6. (Optional) use the **Height** and **Width** fields to change the image size.

7. In the **Clearance Results Preferences** dialog box, click **OK**.

Performing the clearance analysis

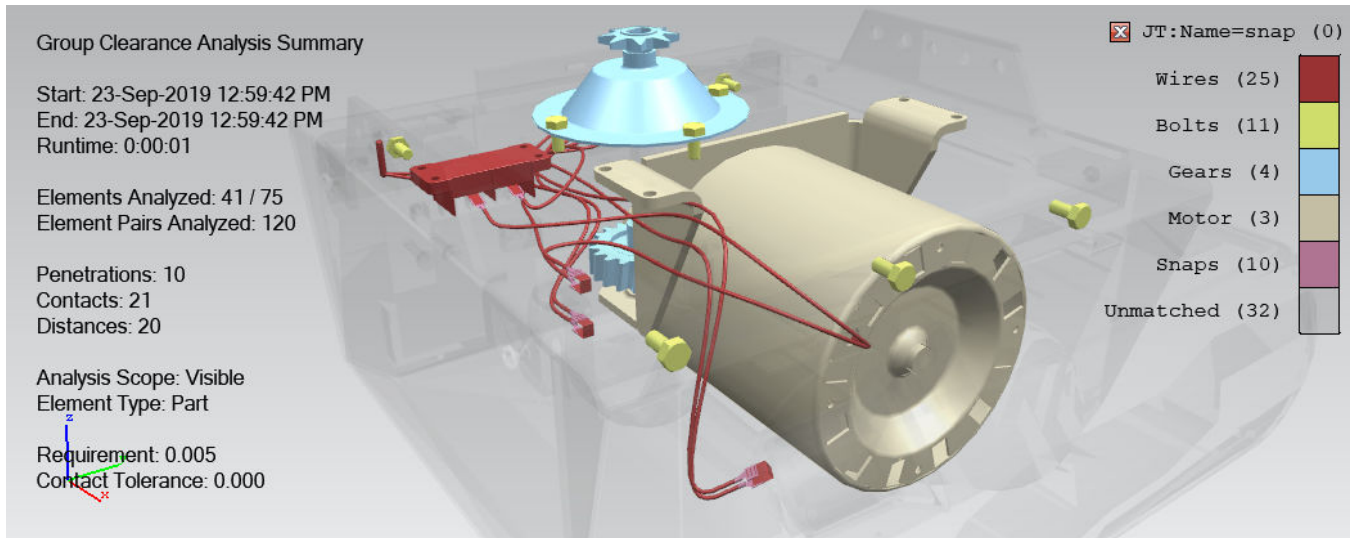
Overview of performing the clearance analysis

Perform clearance analysis to identify spatial conflicts at any stage of the design process. You can check the clearance requirement between both static and moving elements. Conflicts are displayed in a textual list which you can filter to help emphasize critical issues.

Clearance batch analysis can be performed outside of Lifecycle Visualization Mockup. For more information, see Introduction to performing clearance batch analysis in *ClearanceDB — Administration*.

Clearance analysis consists of the following activities:

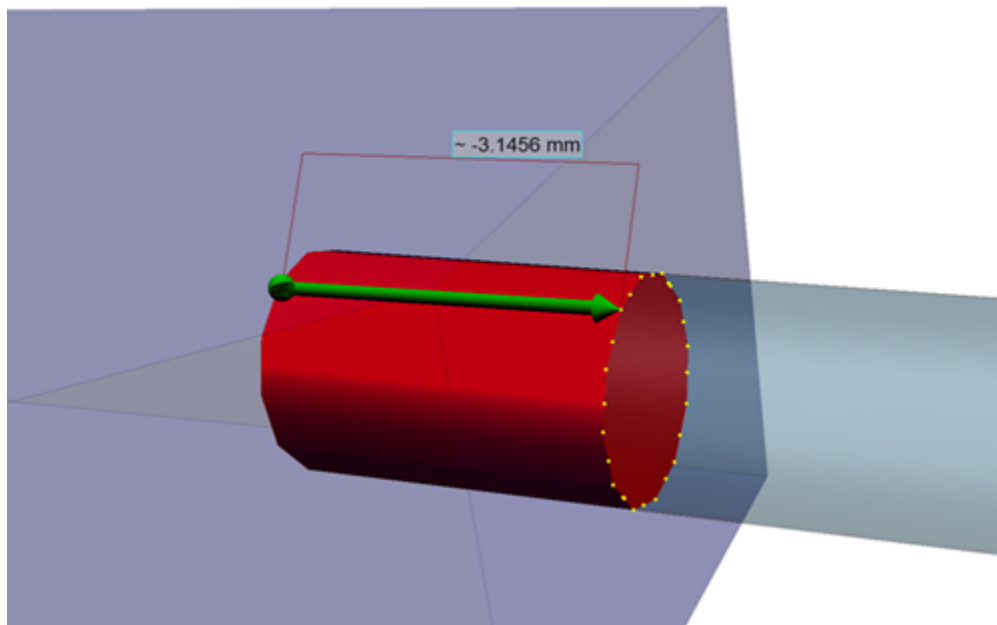
- Understanding the factors involved in clearance calculations and the identification of clearance results.
- Performing clearance checks in four different ways:
 - **Matrix clearance checks** identify non-moving element pairs that are closer to one another than the specified clearance requirement value.
 - **Dynamic clearance checks** identify moving parts that come closer to a selected part than the specified clearance requirement value.
 - **Group to group clearance checks** identify part groups that are closer to one another than the specified clearance requirement value. Group clearance analysis considers groups as a whole, disregarding the distance between parts within the same group or group set.
 - **Selected parts clearance checks** identify parts that come within the clearance requirement of selected parts.
- Understanding the types of clearance violations that are identified with clearance analysis.
- Working with the **clearance results list** to evaluate and manage clearance issues.
- Saving clearance results as a **report**.
- Using a **saved clearance results list**.
- Creating **attribute filters** from part metadata to control information displayed in the **Results** list.
- Analyzing multiple **regions of contact or penetration** between element pairs.



Understanding penetration depth

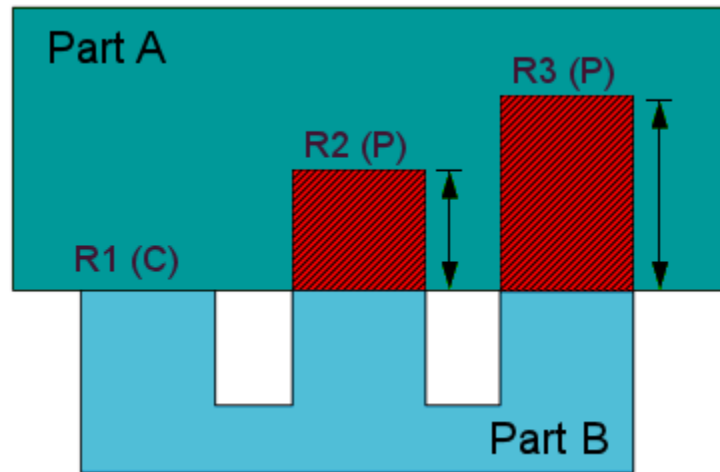
What is penetration depth?

In general, the penetration depth (p-depth) for any two elements (in clearance analysis, the term *elements* is used instead of *parts* to include leaf components and end items) is the distance that one or both elements must be moved in order to resolve the penetration between them. The direction and distance of the transformation are identified by an extraction vector, shown in the example below in green. The magnitude of the extraction vector is the penetration depth.



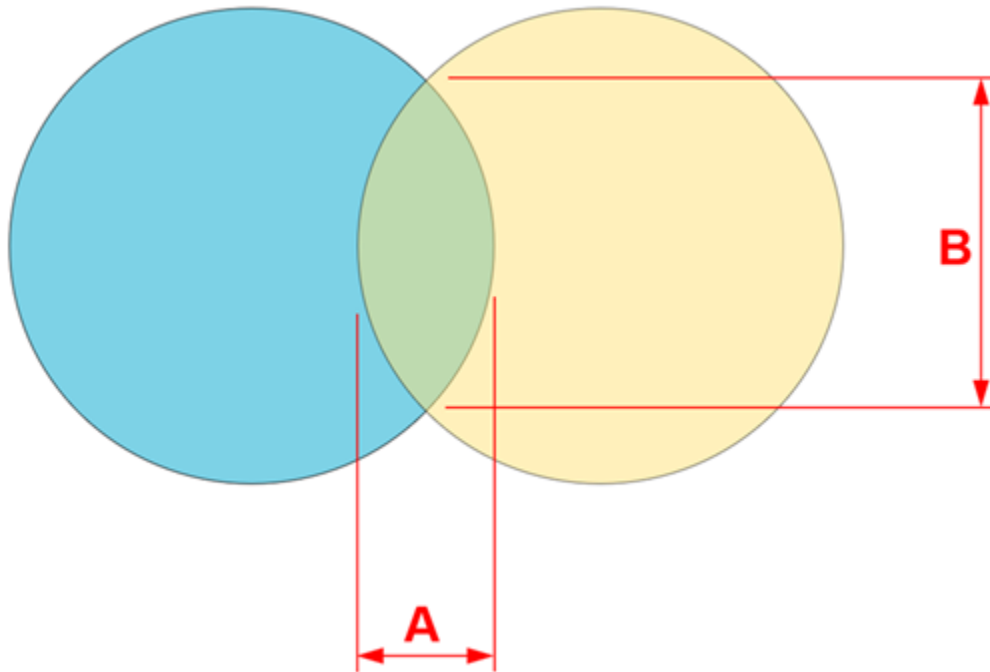
Interference regions

Element pairs can interfere in more than one location, as in the example below. These locations are called regions, and can be either contacts or penetrations. Clearance results of type D (distance) have no regions. If an interfering element pair has more than one penetration region, the overall result reported for that pair is the largest of the depths for all of the regions. This example has a single contact region (R1), and two penetration regions (R2 and R3). The result reported for this pair would be the depth for R3.



No single correct p-depth for any element pair

In the example below, both penetration depths are correct (as well as any results in-between). Clearance penetration depth is not the same as the surface area or volume of a part where there is always exactly one correct answer. For any two penetrating elements there are always an infinite number of valid penetration depths, depending on the direction that is used to separate the two elements.



Penetration depth is an approximation

Clearance analysis in Lifecycle Visualization guarantees to find all violating element pairs within the data being analyzed. However, the p-depth is an *approximation* based on tessellated data, and is not guaranteed to find any perceived correct p-depth or the single-most severe penetration region for any element pair. Only a qualified human with knowledge of the product data can fully understand the severity and magnitude of a penetrating element pair.

The goal of clearance analysis in Lifecycle Visualization is to present users with the list of penetrating element pairs and regions and provide enough information so that a human can best decide how to resolve them.

Penetration depth and the use of NURBS

Penetration depth calculations are always an approximation—even when using precise NURBS data. Selecting the **Use NURBS** preference on the **Clearance Calculator Settings** dialog box does create more accurate contact-penetration points for the two interfering elements, and thus provides more accurate inputs to the p-depth algorithm, but the calculation is still based on tessellation. This is why the p-depth result contains a tilde (~) to reflect the fact that the final response is an approximation.

Consistency with NX

Clearance analyses in Lifecycle Visualization and NX will generally produce very consistent results, however, the code in the two products is not exactly the same. In addition, different preference settings, such as contact and penetration tolerances, can cause slightly different results between the two products. For that reason, exact consistency in every case is not guaranteed.

Analysis types

Matrix clearance analysis

Use **matrix** clearance to check all parts within the analysis scope against all other parts within the analysis scope.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. (Optional) Complete any of the following steps:
 - Set the **clearance requirement**.
 - Specify the **clearance calculator settings**.
 - Change the **precision** of clearance requirement calculations.
 - On the **3D Clearance** toolbar, specify the **scope** of the clearance analysis.
 - Specify the **visibility action** involved in clearance issues.
3. On the **3D Clearance** toolbar, click **Matrix Clearance** .

The **Clearance Analysis** progress bar appears. When the analysis is complete, the **Clearance Results** window is displayed and shows a list of violating element pairs.

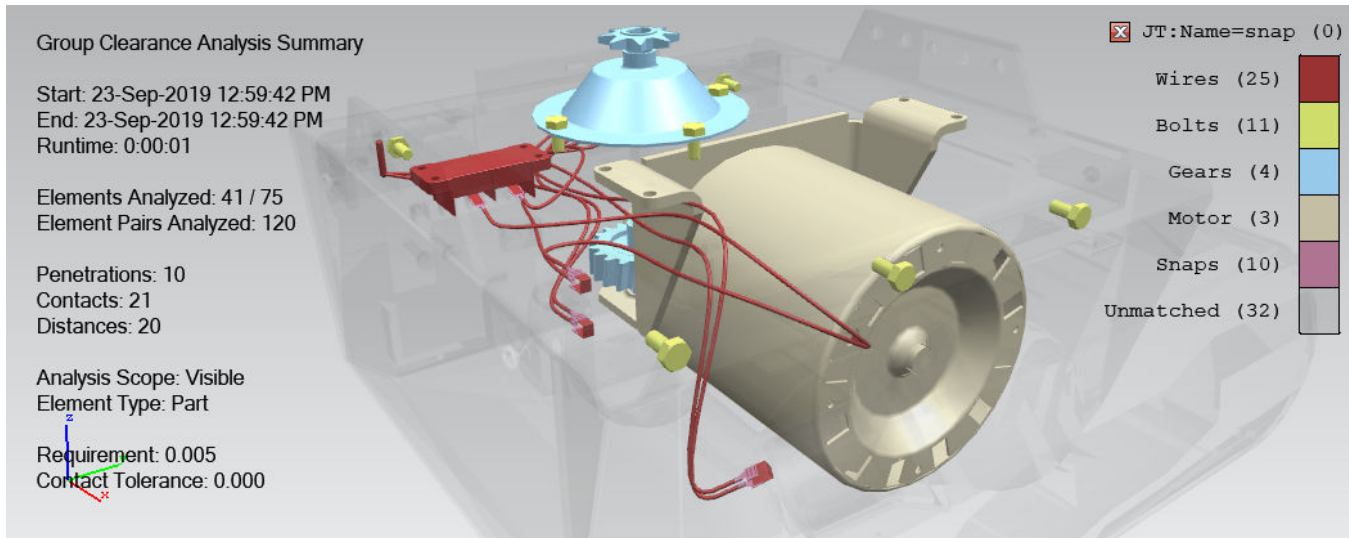
Tip:

To stop generating clearance results, click **Cancel**.

Group clearance analysis

Use **Group** Clearance to check all parts within one group against parts within a second group. It can also check for issues between two sets of groups.





1. Create two or more **groups**.
2. On the **3D Clearance** toolbar, click **Clearance**
3. (Optional) Set the **clearance requirement**.
4. (Optional) Specify the **calculator settings** to perform a contact/penetration check.
5. Choose **Clearance**→**Group to Group Clearance**.
6. In the **Group to Group Clearance** dialog box, under **Group Set 1**, select the groups that you want for the first set of groups.
7. (Optional for **Group Set 1**) Click **Visual Report** to see which parts are included in each group or group set before running the analysis. Select **Combine Groups** to combine the selected groups into one group set for Visual Reports.
8. Under **Group Set 2**, select the groups that you want for the second set of groups.

Note:

In order to analyze each group against every other group, select all groups in both the first and second sets. Lifecycle Visualization Clearance will automatically prevent any groups from being analyzed against themselves. For example, even if group A is in both the first and second group sets, no results between elements in group A will be included.

	A	B	C	D
A	X	O	O	O
B		X	O	O
C			X	O
D				X

9. (Optional for **Group Set 2**) Click **Visual Report** to see which parts are included in each group or group set before running the analysis. Select **Combine Groups** to combine the selected groups into one group set for Visual Reports.
10. In the **Calculation options for each group-group pair** section, select one of the following options:
 - **Report all clearance violations**
 - **Report single worst clearance violation only**
11. (Optional) In the **After the analysis is complete** section, select **Perform Visual Report** to see the colorize results displayed on the Viewing window. Select from the following result options:
 - **Result Types** — shows each result type by color.
 - **Group Sets** — shows each group result by color.

Note:

- To display the visual report legend in the Viewing window, on the **3D Visual Report** toolbar, click **Preferences** , and on the **Legend** tab of the **Visual Report Preferences** dialog, select the **Show Legend** option.
- The counts in the Visual Report legend for **Result Type** show the number of pairs (results), not the number of parts. It is possible for a single part to be included in many clearance results, each with a different type. Since Visual Reports can only give each part a single color, the color assignments are made in this order: penetration (P), contact (C), and distance (D). Therefore, if a single part participates in a P, C, and D result, the color of the part in the report will be the penetration color.

12. Click **Apply** or **OK**.

Results are displayed that identify the parts from the group sets in closest approach or maximum penetration, group names, distances between groups, as well as other information about the part pair.

Note:

- Nothing is displayed if the part pairs between the group sets do not violate the specified clearance.
- Group names are saved to the clearance results file, but if you want to save the group definition you must save the session file.

Selected clearance analysis

Use **Selected** clearance to check all selected parts against all non-selected parts within the analysis scope.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. (Optional) Complete any of the following steps:
 - Set the **clearance requirement**.
 - Specify the **clearance calculator settings**.

- Change the **precision** of clearance requirement calculations.
 - On the **3D Clearance** toolbar, specify the **scope** of the clearance analysis.
 - Specify the **visibility action** involved in clearance issues.
3. **Select** a part.
 4. On the **3D Clearance** toolbar, click **Selected Clearance** .

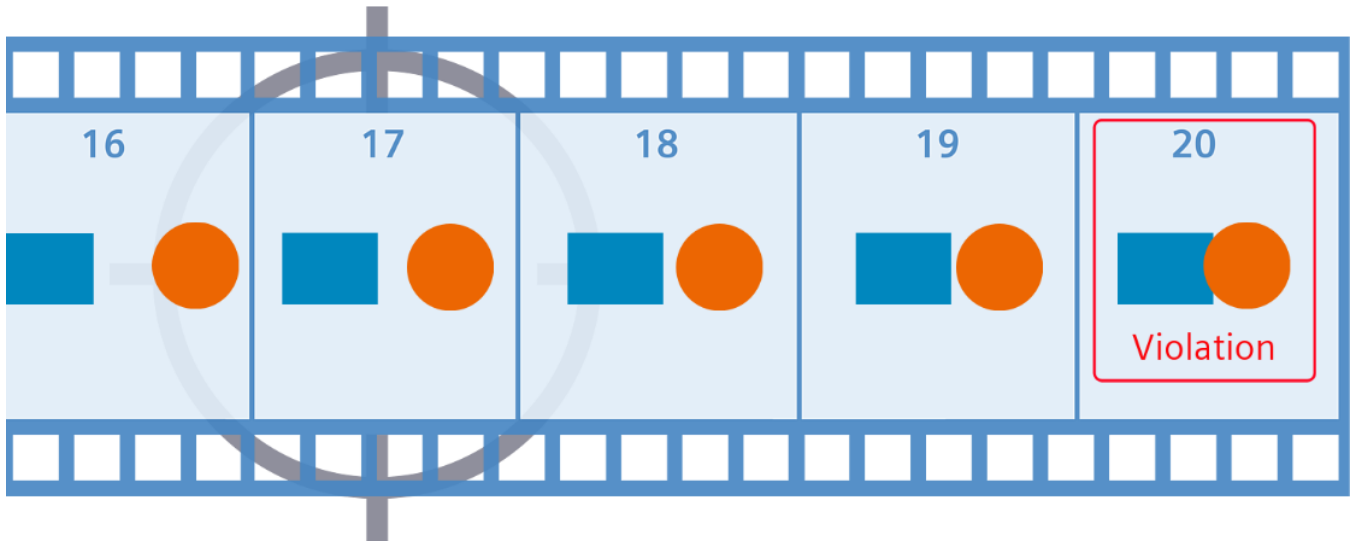
The **Clearance Analysis** progress bar appears. When the analysis is complete, the **Clearance Results** window is displayed and shows a list of violating element pairs.

Tip:

To stop generating clearance results, click **Cancel**.

Dynamic clearance analysis

Use **Dynamic** clearance analysis to identify clearance violations that occur *over time* by using a motion file or while transforming parts in the 3D Viewing window. For example, a violation might be identified between two parts in frame 20 of a motion file.



To locate violations quickly, you can highlight parts that move within the clearance requirement of a selected part.

Note:

- When you use a motion file to perform a dynamic clearance check, a reference to the file is included with the results list when you save it. When you open the list during another work session, double-click a result to open the motion file.
- For results files loaded from remote data repositories, the motion files also must be stored in the remote data repository.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **Clearance** toolbar, click **Dynamic Clearance** .
3. In the **Dynamic Clearance** dialog box, specify the following preferences for dynamic clearance checks:

In this section	Use this option	To do this
Highlighting of Parts in Violation	Highlight Parts in Violation	In the 3D Viewing window, highlight parts involved in clearance issues during motion playback and part transformation.
	Highlight Color Tip: Click the color well to change the highlight color.	Set the color to highlight parts in the 3D Viewing window that are involved in clearance issues.
Analysis Scope	Matrix	Analyze all parts during each frame of motion. This severely affects the motion playback performance.
	Selected Note: You must select the parts for analysis.	Analyze selected parts for violations with non-selected parts during each frame of motion. This significantly affects the motion playback performance.
Results Options (for file-based motion only)	Generate Results List	Create a list of the clearance results.

		Note: A results list is generated only while a motion file is playing. Transforming parts only highlights violations.
	Merge incoming results with existing list	When you create a new list, compare the results against the previous list. List items that appear on the previous list but not the current list are marked as fixed.
	Set as default, do not show this dialog again until requested in Clearance Results Preferences	Stores the selected dynamic clearance preferences as the default values.

4. Click **Enable**.

Note:

When **Analysis Scope** is **Selected**, you must select a part before you can enable the clearance check. You can change your selection after enabling dynamic clearance and this dialog box closes.

5. (Optional) Choose **Clearance** → **Stop on Collision**.

Note:

Use **Stop on Collision** if you are using a motion file and want to stop playback when the parts move within the clearance requirement of other parts. Note however that the clearance system can check for collisions only once per motion frame. The farther a moving part travels between frames, the larger the potential penetration is if a collision occurs.

6. (When **Analysis Scope** is **Selected**) In the 3D Viewing window, select a part.

Parts within the clearance requirement of the selected part are highlighted.

7. Do one of the following:

- Load and play a **.vfm motion file**.
- **Transform** parts of your model.

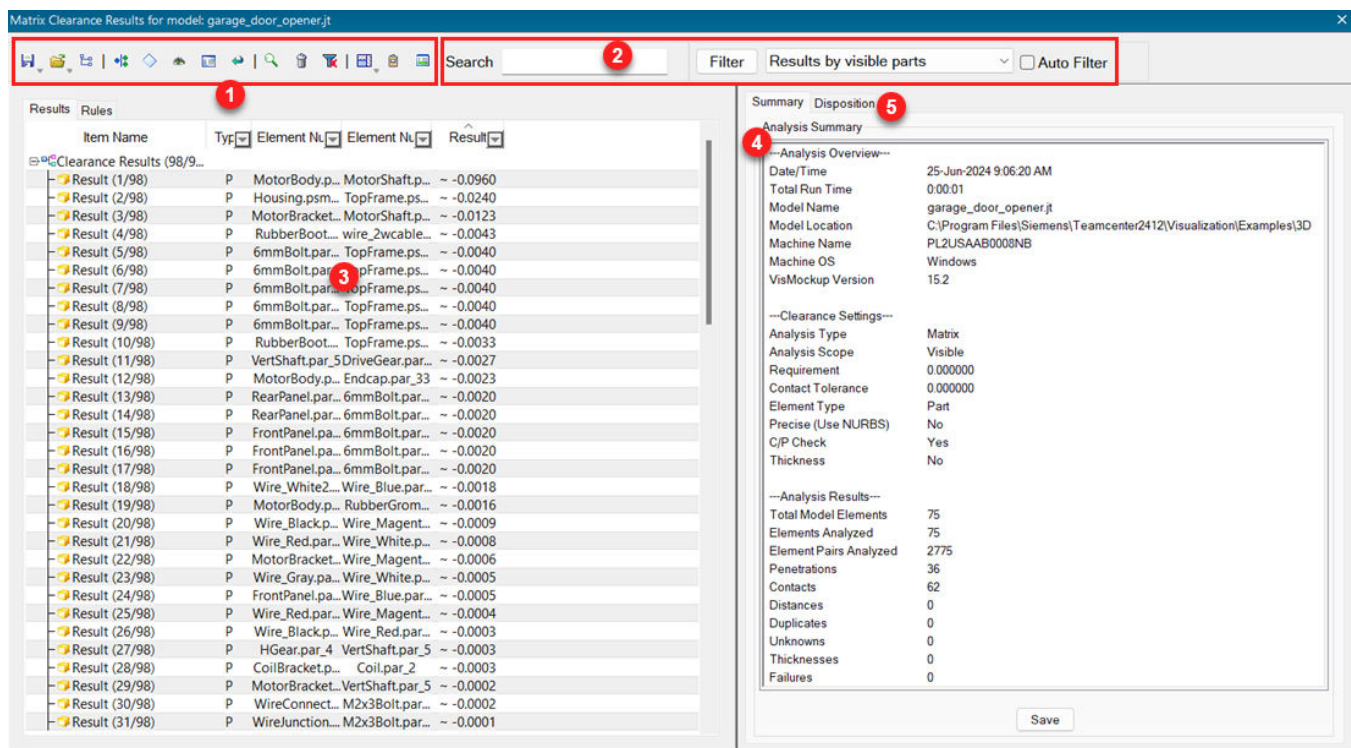
Depending upon the dynamic clearance preferences, parts that move within the clearance requirement of the selected part may be highlighted. A clearance results list may be generated for file-based motion.

8. On the **Clearance** toolbar, click **Dynamic Clearance** , so it is deactivated.

Working with the analysis results

About clearance analysis results

When you perform clearance analysis, you work with the clearance results list to evaluate and manage clearance issues.



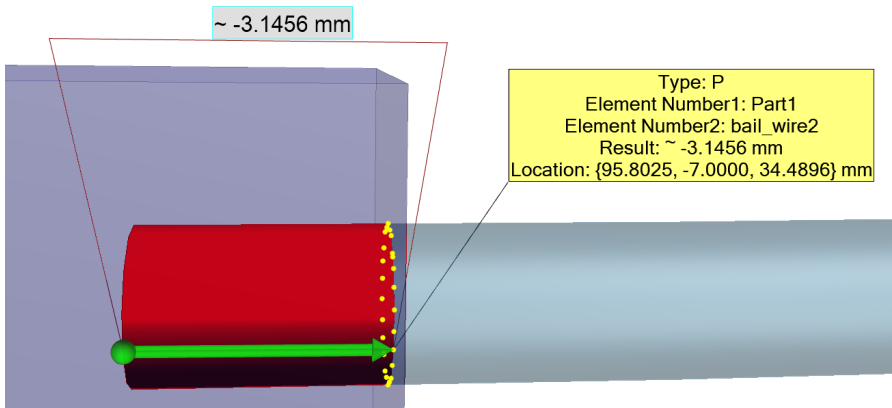
- 1 Use the **Clearance Results toolbar** to work with the clearance results list, including, saving, loading, filtering, searching, and customizing the way results appear.
- 2 **Filter the Results list** by using a search term or by choosing to match selected or visible parts.
- 3 Use three **shortcut menus** to perform many tasks in the **Clearance Results** window.
- 4 Use the **Summary tab** to view a summary of clearance analysis.
- 5 Use the **Disposition tab** to specify the disposition of clearance issues.

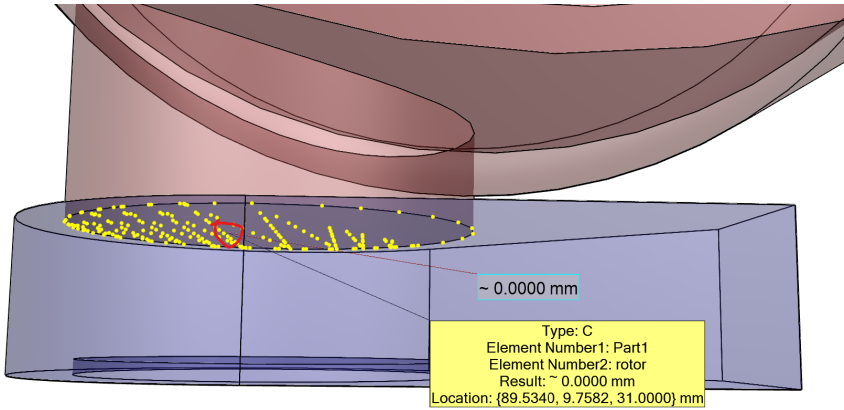
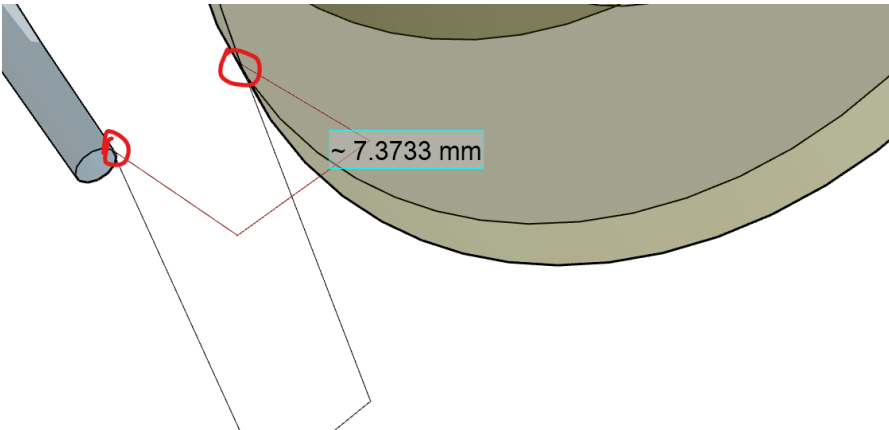
Overview of the clearance results list

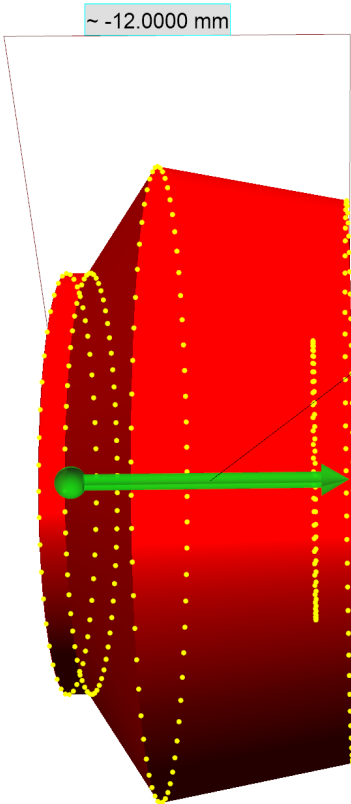
The clearance results list appears on the Results page of the **Clearance Results** window when you perform a clearance check. The window displays a list of the element pairs that violate the specified clearance requirement.

In addition to the four columns that appear in the list by default, you can display a column for any attribute that is included in your data. You also can remove or reorder the columns.

This column	Displays this information
Type	The type of clearance between the elements. Note: <i>What is Clearance?</i> provides more details about clearance types. C (Contact) - A hard interference. Elements are in contact but do not penetrate. D (Distance) - A soft interference. Dup (Duplicate) - Elements have identical geometry and are located directly on top of each other. P (Penetration) - A hard interference. Elements penetrate each another. T (Thickness) - A hard or soft interference. Elements conflict using material thickness. U (Unknown) - A hard interference. Further analysis is required to classify as a contact or penetration. Right-click to analyze clearance or enable contact/penetration checking. Note: Element pairs involving thickened sheets may have clearance issues classified only as T when the material thickness is taken into account. For more accurate analysis, the resultant clearance types may be D or U type for distance or contact/penetration relationships, respectively. No contact/penetration or distance line analysis is available for such element pairs, such as ones involving thickened sheets. Tip: You can set clearance results preferences to display clearance type descriptions in the Results list.
Element Number1	The name of the first element of the pair.

This column	Displays this information
Element Number2	The name of the second element of the pair.
Result	The distance between the elements.
Requirement	The clearance requirement you specified.
Location	(If you selected Distinguish between Contact and Penetrations in the Clearance Calculator Settings dialog box) The average location and the average normal of the point of contact between the element pairs. The location numbers have different meanings for each result type. Tip: It may be useful to enable the Show annotations in the 3D view when displaying issues preference when viewing clearance issues. Penetration — the 6 values represent the location of the result and the direction of the penetration vector. The value in the Result column is the magnitude of the penetration vector. The values in the Location column represent the point in 3D space where the clearance violation occurred.  Contact — the 6 values in the Location column represent the geometric center of the contact points, and the normal to the best fit plane containing the points.

This column	Displays this information
	 ~ 0.0000 mm Type: C Element Number1: Part1 Element Number2: rotor Result: ~ 0.0000 mm Location: {89.5340, 9.7582, 31.0000} mm • Distance — the 6 values in the Location column identify the locations of the closest points between the two parts.  ~ 7.3733 mm Type: D Element Number1: spool Element Number2: bail_wire2 Result: ~ 7.3733 mm Location: {99.0000, -2.7544, 25.8382} {96.5386, -3.9100, 32.6919} mm • Duplicate — the 6 values in the Location column are the center of the bounding box containing the duplicate part pair and the direction of the penetration vector (the same as for penetrations).

This column	Displays this information
	 ~ -12.0000 mm Type: Dup Element Number1: drag_knob Element Number2: drag_knob Result: ~ -12.0000 mm Location: {8.2215, -0.0000, -17.7508} mm Note: When you analyze subsets in the viewer that were transformed in Teamcenter (using Translate or Rotate) and then load pre-calculated results from the ClearanceDB database, the locations of the results in the viewer are different than those in the database. Therefore, the locations of the ClearanceDB results are adjusted to reflect the subset transforms as they are loaded.
Frame	(If you selected Generate Results List in the Dynamic Preferences dialog box) The result of the current new frame.
Intersection Volume	(If you selected Calculate in the Intersection Volume in the Clearance Calculator Settings dialog box and Precise in the Intersection Volume section of the Clearance Results Preferences dialog box) The intersection volume between any two penetrating elements involved in a clearance violation.

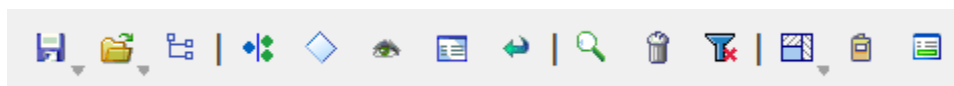
Note:

While **analyzing clearance while a model is in motion**,

- An element pair may appear more than once in the results list with different frame numbers. This would be the case if an element pair violates the user-specified clearance in more than one frame of the motion run.
- If an element pair does not violate clearance in a certain motion frame, the pair will not appear in the list for that frame. Also, if one element of the element pairs becomes out of the scope against which the clearance is checked at some frame, the pair will not appear in the list for that frame even though there may be violations. This is because the list only shows violations to the clearance criteria set up by the user.
- A new results list is generated each time dynamic clearance analysis is turned on. While dynamic clearance analysis is on, results are appended to the results list during motion playback. This results list contains the frame numbers associated with the motion file for each result. The main use case for generating results is to select elements to check clearance against and then play the motion file. While it is possible to change the selected elements or transform elements while the motion file is playing/paused, or skip frames during playback, doing so may result in an ambiguous results list.

Clearance Results toolbar

On the **Results** tab in the **Clearance Results** window, use the toolbar to work with the clearance results list, including: saving, loading, filtering, searching, and customizing the way results appear.



	Use this option	To do this
	Save Results (Save All or Save Visible)	Save the results list as a .txt file or Clearance database Connection (.dbc) file.
	Load Results (Browse... or Quick-load)	Choose Browse... to load a saved results list . Choose Quick-load to load the last opened results list
	Matrix Clearance	Perform matrix clearance checks to identify non-moving element pairs that are closer to one another than the specified clearance requirement value.
	Analysis Mode	Enable analysis mode . When this option is activated, clearance issues are displayed in the Viewing window when you select them in the results list.

	Use this option	To do this
	Cross Section	Show or hide a cross section at the region of contact or penetration.
	Analysis Pair Visibility	Adjust the visibility of the element pair, intersection volume, and clearance annotations to improve visualization in the 3D view.
	Open Clearance Navigation Dialog	Open the Clearance Navigation & Analysis dialog box to analyze regions of contact or penetration.
	Clear Analysis in View	Clear the clearance issue that appears in the Viewing window.
	Find in Results	Search for text strings in the results list.
	Clear Results List	Clear the results from the results list.
	Clear All Filters	Clear all applied filters.
	Show Windows (Show Occurrences or Show Toolbox)	Choose Show Occurrences to show the window listing occurrences. This is available only if multiple instances of part pair violations exist in your clearance results. Choose Show toolbox to show the Clearance Toolbox .
	Results Report	Create a report of the clearance results.
	Preferences	Specify preferences for the display of clearance results.

Search and filter clearance results

On the Results page in the **Clearance Results** window, you can filter the results list by using a search term or by choosing to match selected or visible parts.

Filter by using a search term

1. Type a search term in the **Search** box at the top of the **Clearance Results** window.

As you type characters in the **Search** box the results list is filtered.

Note:

Searches performed using the **Search** box are not case-sensitive.

2. Use the Backspace key to remove terms from the **Search** box.

Filter by selected or visible parts

Search and filter the list or the Viewing window to match selected or visible parts.

1. From the **Filter** list at the top of the **Clearance Results** window, select any of the following:

- **Results by visible parts**

List the results that are related to all parts visible in the Viewing window.

- **Results by selected parts**

List the results related to all parts that are selected in the Viewing window.

- **Parts by visible results**

Display the parts related to all results that are currently shown in the results list.

- **Parts by selected results**

Display the parts related to all results that are currently selected in the results list.

2. Click **Filter**.
3. (Optional) To apply the filter automatically each time you change the part selection or visibility state, select the **Auto Filter** check box.

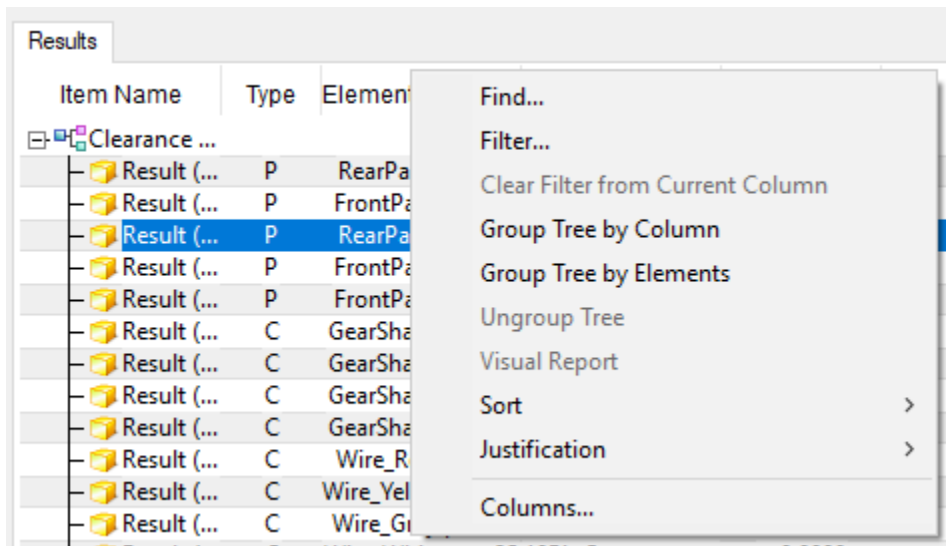
Options for working with the results list

Results list shortcut menus

In the **Clearance Results** window (Results page), you can access shortcut menus by right-clicking in different areas of the window.

Column heading shortcut menu

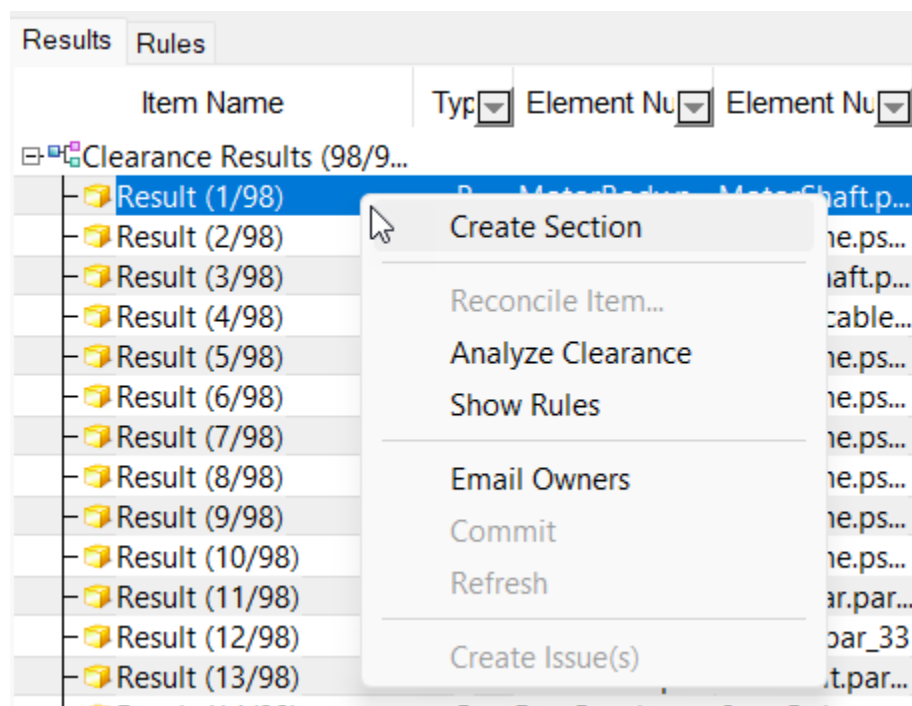
Right-click any column heading to choose these options.



Use this option	To do this
Find	Search for text strings in the Results list.
Filter	Show and hide the column filters . When column filters are shown, you can choose a column filter to filter the list by the information in the column.
Clear Filter From Current Column	Remove applied column filters.
Group Tree by Column	Group the results by column.
Group Tree by Elements	Group the results by element.
Ungroup Tree	Remove a grouping by column or element.
Visual Report	Update the visual report to reflect the selected column.
Sort	Sort the results list by the information in any column, in either ascending or descending order. Note: Click the column heading to switch between an ascending and descending order.
Justification	Align the column text to the Right , Center , or Left , for optimum readability.
Columns	Display a column for any attribute that is included in your data, as well as remove or reorder the columns.

List items shortcut menu

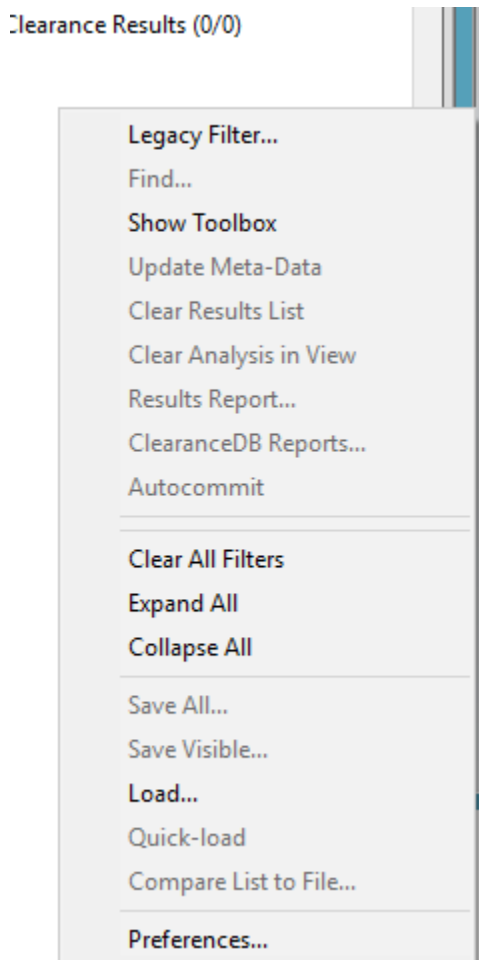
Right-click an item in the **Results** list to choose these options.



Use this option	To do this
Create Section	Manipulate a plane through your model and display a cross section of the model where it is cut by the plane.
Reconcile Item	When comparing current results to a saved results list an element pair that has changed.
Analyze Clearance	Analyze a clearance issue. The issue appears in the Viewing window, and the Clearance Navigation & Analysis reconcile dialog box appears.
Show Rules	Show the rules associated with the selected result. This option is available when analyzing locally or when Clearance rules and results are loaded from either ClearanceDB or from .csvcldb files. (Both the rules and results must be loaded from the same source.)
Email Owners	Notify the owners of a clearance issue by email of a status change.
Commit	Manually update the ClearanceDB database with the revised status of a clearance issue.
Refresh	Check the ClearanceDB database for possible updates to the attribute information displayed in the Results list.
Create Issue(s)	Capture selected clearance violations as visual issues .

Clearance Results window shortcut menu

Right-click in the background of the **Clearance Results** window to choose these options.



Use this option	To do this
Legacy Filter	Filter the results to display only element pairs that meet specified criteria.
Find	Search for text strings in the results list.
Show Toolbox	Show or hide the Clearance Toolbox .
Update Meta-Data	Include metadata with the results, if any is available for the elements and the columns are displayed in the Results list.
Clear Results List	Clear the results from the Results list.
Clear Analysis in View	Clear the clearance issue that appears in the Viewing window.
Results Report	View clearance results using reports
ClearanceDB Reports	View ClearanceDB results using reports.

Use this option	To do this
Autocommit	Automatically update the ClearanceDB database with the revised status of a clearance issue.
Clear All Filters	Remove applied filters.
Expand All	Expand all items in the results list.
Collapse All	Collapse the items in the results list.
Save All	Save the items that appear in the Results list as well as any filtered items.
Save Visible	Save only the items that appear in the Results list.
Load	Load a saved results list.
Quick-load	Automatically load the last opened results list.
Compare List to File	Compare the current results with a saved results list.
Preferences	Specify preferences for the display of clearance results.

Sort, filter, and position columns in the results list

Do the following to sort, filter, and position the columns of the results list in the **Clearance Results** window.

Sort results by column

Do either of the following:

- Click a column heading to switch between sorting it in ascending and descending order.
- Right-click a column heading, and choose **Sort→Ascending** or **Descending**.

If you select a new column to sort the results, the previously selected column becomes a secondary sort.

Filter results by column

1. Do either of the following:

- On the **Clearance Results** toolbar, click **Clear All Filters** .
- Right-click a column heading, and choose **Filter**.

2. Click the arrow to the right of a column name, and then in the filter dialog box that appears select or clear the check boxes.

3. To clear filters, do one of the following:
 - If the column filter dialog box is open, click **Clear Filter from** *column name*.
 - Right-click a column heading, and choose **Clear Filter From Current Column**.
 - Right-click in the background of the **Results** list, and choose **Clear All Filters**, or click **Clear All Filters**  on the **Clearance Results** toolbar.

Group results by column or element

1. Right-click a column heading, and choose one of the following:
 - **Group Tree by Column**
 - **Group Tree by Elements**
2. To remove a grouping, right-click a column heading, and choose **Ungroup Tree**.

Add columns to the results list

1. On the **3D Clearance** toolbar, click **Results Preferences** .
2. On the **Columns** tab of the **Clearance Results Preferences** dialog box, in the **Optional Columns** section, select the attribute you want to include as a new column in the list.
3. Click **Add**.

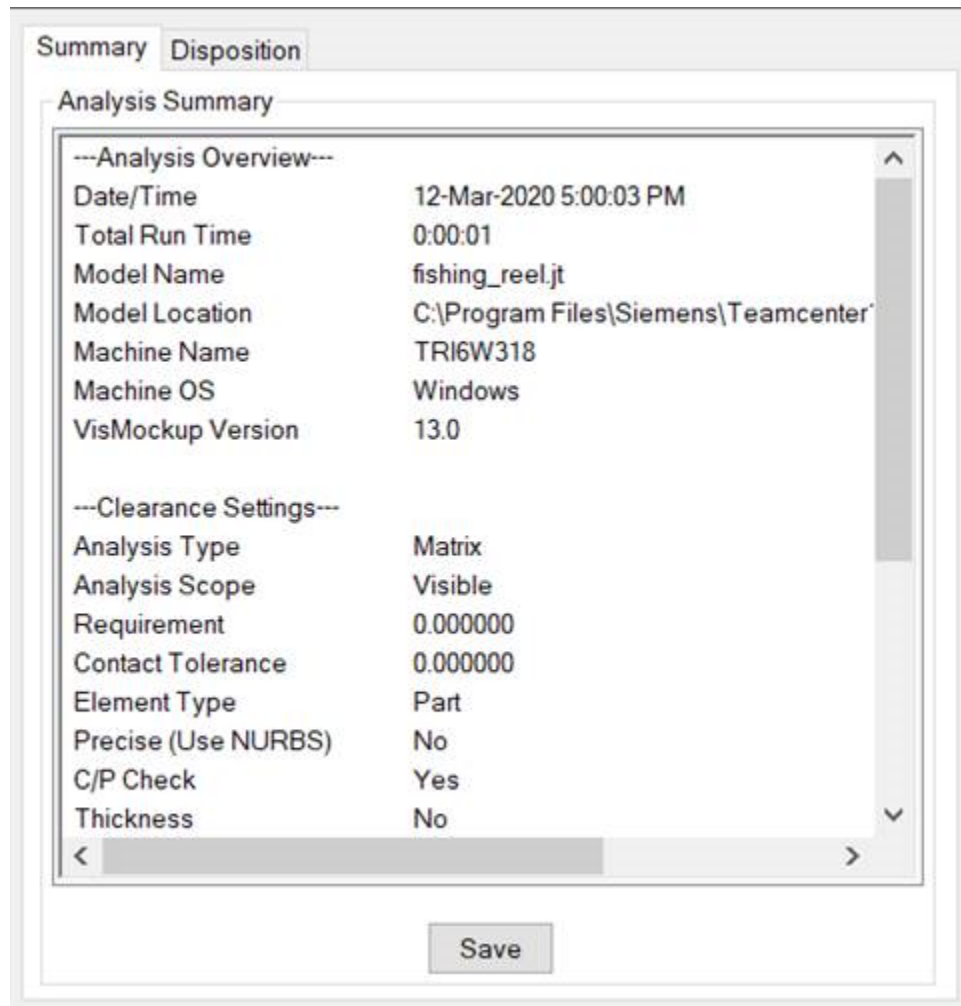
The attribute appears in the **Visible Columns** section.

4. (Optional) In the **Visible Columns** section, select an attribute, and click **Up** or **Down** to rearrange the order of the columns as they appear in the **Results** list.
5. Click **OK**.

Using the results list Toolbox

Summary tab

You can view a summary of a clearance analysis on the **Summary** tab of the **Clearance Toolbox**.



1. On the **Clearance Results** toolbar, click **Dispositions**  to display the **Clearance Toolbox**.

The analysis summary is displayed on the **Summary** tab of the **Clearance Toolbox**.

2. To save the analysis summary as a report, click **Save**.

Disposition tab

You can specify an issue **disposition** for each clearance result, capturing such information as owner, priority, status, and more. You can then display the issue disposition information in the **Results** list.

The screenshot shows a software window with two tabs: 'Summary' and 'Disposition'. The 'Disposition' tab is active. Inside, there's a section titled '1 Issue Selected'. Below this, there are several input fields and checkboxes. 'Owner' and 'Zone' are dropdown menus with 'Add...' buttons next to them. 'Priority' is a dropdown menu set to 'None'. 'Status' is a dropdown menu set to 'Not Evaluated'. There are two checkboxes: 'Important' (checked) and 'Change Required' (unchecked). Next to 'Important' is a 'Comment:' text box. Next to 'Change Required' is a 'Change Reference:' text box. At the bottom, there are three buttons: 'Apply', 'Reset', and 'Commit to Clearance Database'.

Creating clearance filters with element attributes

Introduction to creating clearance filters with element attributes

Create filters using element metadata to limit the information displayed in the **Clearance Results** window. When you apply a filter, results are displayed only if the specified attributes contain values that match the criteria you have specified.

Specify attributes to include in clearance results

Choose the element attribute metadata that you want to include in the **Results** list.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. Choose **Clearance**→**Preferences**→**Attributes**.
3. In the **Attribute Properties** dialog box, in the **Regenerate Attribute List from** section, specify the scope of the elements from which you want to examine for attributes. You can choose from **All**, **Visible**, or **Selected elements**.
4. Click **Regenerate**.

Available attributes appear in a scrollable list, and are based on the first element in the product structure.

5. In the list of available attributes, select the check box for each attribute you want to include in the **Results** list.
6. For each attribute that you choose, specify the search type. Choose from the following search types:

- **Element Only**
- **Element Parent Only**
- **Element and Parents (Element First)**
- **Element and Parents (Root First)**
- **Parents Only (Element Parent First)**
- **Parents Only (Root First)**

7. Select **Adjust column visibility**.

The attributes appear in the **Results** list as Element1-[attribute_name] and Element2-[attribute_name].

8. (Optional) To regenerate the attribute list, click **All**, **Visible**, or **Selected**, and then click **Regenerate**.
9. (Optional) To merge the current attribute list with attributes from an external text file, click **Merge Saved**, browse to the text file, select the file, and click **Merge Saved**.
10. (Optional) To clear the current attribute list, click **Clear**.
11. Click **OK**.

Define a filter value

Specify a value for the filter to compare with the attribute value. If the attribute value matches the specified parameter, the element pair passes the filter and is displayed in the **Results** list.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. In the **Clearance Results** window, right-click and choose **Filter**.
3. In the **Results Filter** dialog box, click **Add**.
4. In the **Filter Values** dialog box, from the **Select Column** list, select the column heading that contains the information you want to filter.
5. From the **Select Column Data Type** list, specify the search type. Select **Text** to search for text strings, or **Number** to search for numeric values.
6. (If filtering for number values only) From the **Select Filter Operator** list, choose the operator that describes the number relative to the value.

- From the **Value** list, type the text string or numeric value that you want to filter.

Example:

To filter all the element pairs with a clearance requirement that is less than or equal to .02, complete the dialog box as follows:

Select Column: Result

Select Column Data Type: Number

Select Filter Operator: <=

Value: 0.02

- Click **OK**.

The column and value appear in the **Filter Sequence** list.

Specify the input and results of the filter

Specify which element pairs are subject to the filter and what happens to those element pairs that pass the filter.

- On the **3D Clearance** toolbar, click **Clearance** .
- In the **Clearance Results** window, right-click in the background, and choose **Legacy Filter**.
- In the **Results Filter** dialog box, in the **Filter Input** section, click **All Results** to use the filter on the entire results list, or click **Visible Results** to use the filter only on the list items that are currently visible.

Tip:

When you create a series of filters, select **Visible Results** to use the output of the previous filter as input for the selected filter.

- From the **Filter Action** list, select an option for the results of the filter.

Use this option	For this
None	Disable the selected filter.
Show Only	Display the rows in the list that pass the filter, and hide all other rows.
Show Additive	Display the rows in the list that pass the filter in addition to the rows that are currently displayed.
Hide	Hide the rows that pass the filter.

Use this option	For this
Select Only	Select the rows that pass the filter, and deselect all other rows.
Select Additive	Select the rows in the list that pass the filter in addition to the rows that are currently selected.
Deselect	Deselect the rows in the list that pass the filter.

5. In the **Filter Scope** section, under **Element Range**, choose one of the following options:

Use this option	For this
No Match Required	All clearance issues. No filter scope is applied.
All Elements	Clearance issues with node keys that can be matched with element pairs or that are present in the database.
Visible Elements	Clearance issues that can be matched with visible element pairs in the current view.
Selected Elements	Clearance issues that can be matched with selected element pairs in the current view.

6. From the **Matching Element(s)** list, choose one of the following options:

Use this option	For this
Both	Both elements associated with a clearance issue must match the filter criteria to be included in the results.
Either	Either element associated with a clearance issue must match the filter criteria to be included in the results.

When you apply the filter, it is applied to all or only visible element pairs, as you have specified. The element pairs that pass the filter are also displayed or selected as you have specified.

Apply a filter

- On the **3D Clearance** toolbar, click **Clearance** .
- In the **Clearance Results** window, right-click and choose **Filter**.
- Click **Apply**.

Managing filters

Save and load multiple filters to control the display of results in the **Results** list.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. In the **Clearance Results** window, right-click and choose **Filter**.
3. In the **Results Filter** dialog box, click **Manage**.
4. Do any of the following:

To	Do this
Save the current filter	a. Click Add. A new entry appears in the Filter Names list. b. Type a name for the filter, and press Enter. c. Click Save.
Load a saved filter	a. Click Load. b. In the Load Filter dialog box, select a filter, and click OK.
Select a saved filter	In the Filter Names list, select a filter, and click OK .
Delete a filter	In the Filter Names list, select a filter, and click Remove .
Reorder the Filter Names list	In the Filter Names list, select a filter, and click Up or Down .
Rename a filter	a. In the Filter Names list, double-click a filter. b. Type a new name for the filter, and press Enter.
Specify to automatically apply the current filter if the clearance results change	Select the Apply filters to new or changed list items check box.
Specify a default filter	Select the Load the current filter on startup check box.
Save the current filter as a .reg file	a. Click Advanced. b. In the REG File section, click Export. c. Enter a name for the registry file, browse to the location where you want to save the file, and click Save.
Load a filter from a .reg file	a. Click Advanced. b. In the REG File section, click Import.

To	Do this
	c. Browse to the location of the file you want to load, select the file, and click Open .

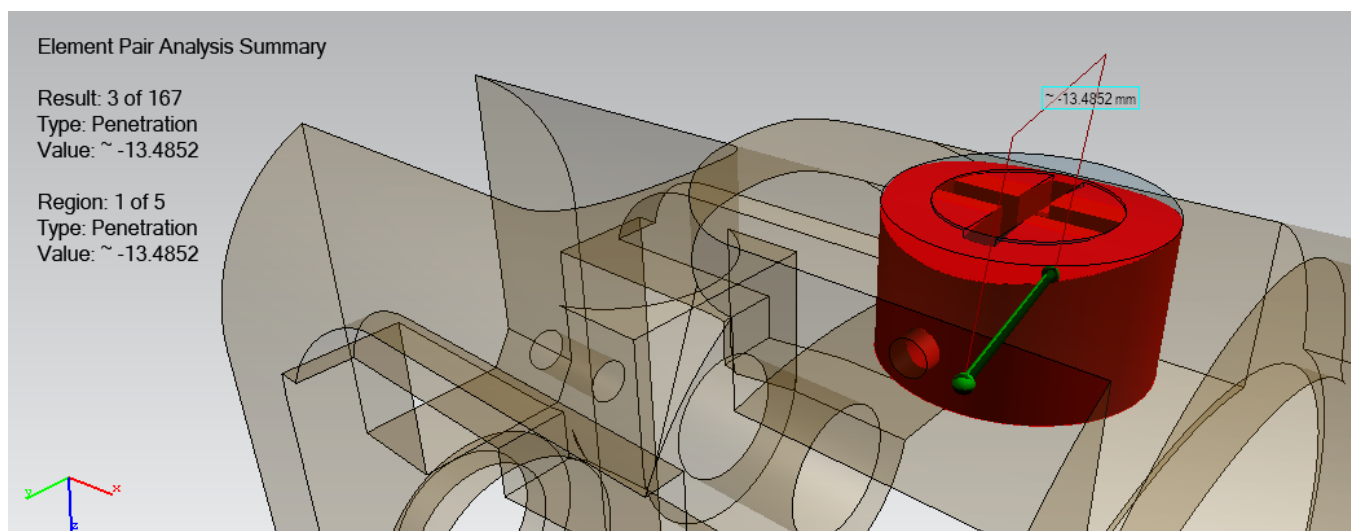
5. Click **OK**.

Analyzing violating element pairs

Overview of analyzing violating element pairs

When you perform a clearance analysis, the output of the analysis is a list of violating element pairs called *results*, or *violations*. Each of those violations can then be examined in detail in a process called *analyzing clearance violations*. During that process, you can do the following.

- Customize the **default appearance** of the elements involved in the violation.
- Analyze regions of **contact or penetration**.
- Display **intersection volumes**.
- Display a **penetration vector** to show the magnitude and direction of the penetration between two elements.
- Display **distance lines** between elements that are closer to one another than the specified clearance requirement.
- Display a **cross section** through the points of contact or penetration.
- Specify **clearance results preferences**.



Display clearance violations in the Viewing window

By default, when you choose to display a clearance issue, it is analyzed according to the settings in the **Clearance Results Preferences**.

In addition to displaying clearance violations, you can **further analyze the contact region**.

Display element pairs in a clearance issue

Double-click a clearance issue.

In the Viewing window, the elements are displayed.

Perform a single click analysis

1. On the **Clearance Results** toolbar, click **Analysis Mode** .
2. Click a clearance issue.

Display a cross section at the region of contact or penetration

You can manipulate a plane through your model and display a **cross section** of the model where it is cut by the plane.

1. Do either of the following:
 - Right-click the element pair and choose **Create Section**.
 - On the **Clearance Results** toolbar, click **Cross Section** .
2. Do one of the following to remove the cross section:
 - Right-click in the results list, and choose **Create Section** to clear the check mark.
 - On the **Clearance Results** toolbar, click **Cross Section** .

Clear an analysis in the Viewing window

Do any of the following:

- Right-click a clearance issue or an empty area of the **Clearance Results** window, and choose **Clear Analysis in View**.

- On the **3D Clearance** toolbar, click **Clear Analysis in View** .

Configure the element pair display

Change the default appearance of elements in the Viewing window when a clearance result is selected from the **Results** list or the **Clearance Navigation & Analysis** dialog box. Categories include each element in an element pair, the intersection volume between the two elements, and all other elements.

- On the **3D Clearance** toolbar, click **Clearance** .
- On the **3D Clearance** toolbar, click **Results Preferences** .
- On the Element Display page of the **Clearance Results Preferences** dialog box, select one of the following element categories:

Select this category	To do this
Element1	Customize the appearance of the first element of a selected element pair.
Element2	Customize the appearance of the second element of a selected element pair.
Visible Elements	Customize the appearance of other visible elements when an element pair is selected.
Intersection Volume	Customize the appearance of the area where Element1 and Element2 intersect.

- Customize the selected element category in one or more of the following ways:

To	Do this
Change the material color	- Select Shading . (Element1 and Element2 categories only) Deactivate Material Color. - Choose a different color from the color well in the Shaded box. When you generate a list, the material color will apply to the element category in the Viewing window.
Change the transparency	- Select Shading . - Select Transparency.

To	Do this				
	c. Do one of the following: - Click and drag the Transparency slider toward Transparent to make the element more transparent. - Click and drag the slider toward Opaque to make the element more solid. - Deselect the Transparency check box and disable transparency to make the element 100% solid.				
Customize the appearance of the tessellation lines	a. Select Tessellation Lines . Note: - You cannot apply tessellation lines to an Intersection Volume category. - The Tessellation Lines option is not editable for Element1 and Element2 categories unless you select the Shaded option. - The Tessellation Lines button may be disabled if both Shaded  and Feature Edges  are selected. To enable Tessellation Lines, deselect Feature Edges and then select Tessellation Lines. b. Do one or both of the following: - Select a different color from the color well. - Select one of these options to define the appearance of tessellation lines on overlapping elements: <table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="text-align: left;">Use this option</th><th style="text-align: left;">To do this</th></tr> </thead> <tbody> <tr> <td>Invisible </td><td>Make tessellation lines that are behind other lines invisible.</td></tr> </tbody> </table>	Use this option	To do this	Invisible 	Make tessellation lines that are behind other lines invisible.
Use this option	To do this				
Invisible 	Make tessellation lines that are behind other lines invisible.				

To	Do this
	Ghosted  Make tessellation lines that are behind other lines appear dashed.
	Visible  Make tessellation lines that are behind other lines entirely visible.
Display feature edges	a. Select Feature Edges  . Note: - You cannot apply feature edges to an Intersection Volume category. - The Tessellation Lines option is not editable for Element1 and Element2 categories unless you select the Shaded option.

- (Optional) Enable **Set As Default** to apply your settings not only to your current session but to all future sessions.
- (Optional) Click **Reset** to revert to the default settings for an element category.

The Element Display page reverts to the element category's default settings.

- Click **OK**.

Adjust the visibility for a specific element pair

So that you can more easily interpret the results of a specific element pair, use the following procedure to adjust the transparency of the elements and intersection volume during pair analysis. These settings override the default values defined in the **Clearance Results Preferences** dialog.

- On the **3D Clearance** toolbar, click **Clearance** .
- On the **3D Clearance** toolbar, click **Results Window** .
- Perform a clearance analysis.
- In the **Results** list, right-click a clearance result and choose **Analyze Clearance** from the menu.

-or-

Double-click a clearance result to begin the analysis.

5. On the **Clearance Navigation & Analysis** dialog, select the desired options and click **OK**.

This dialog displays automatically when you right-click and select **Analyze Clearance** from the menu. However, when you double-click a clearance result, this dialog displays only when the **Show Dialog** checkbox is selected on the **Analysis** page on the **Clearance Results Preferences** dialog.

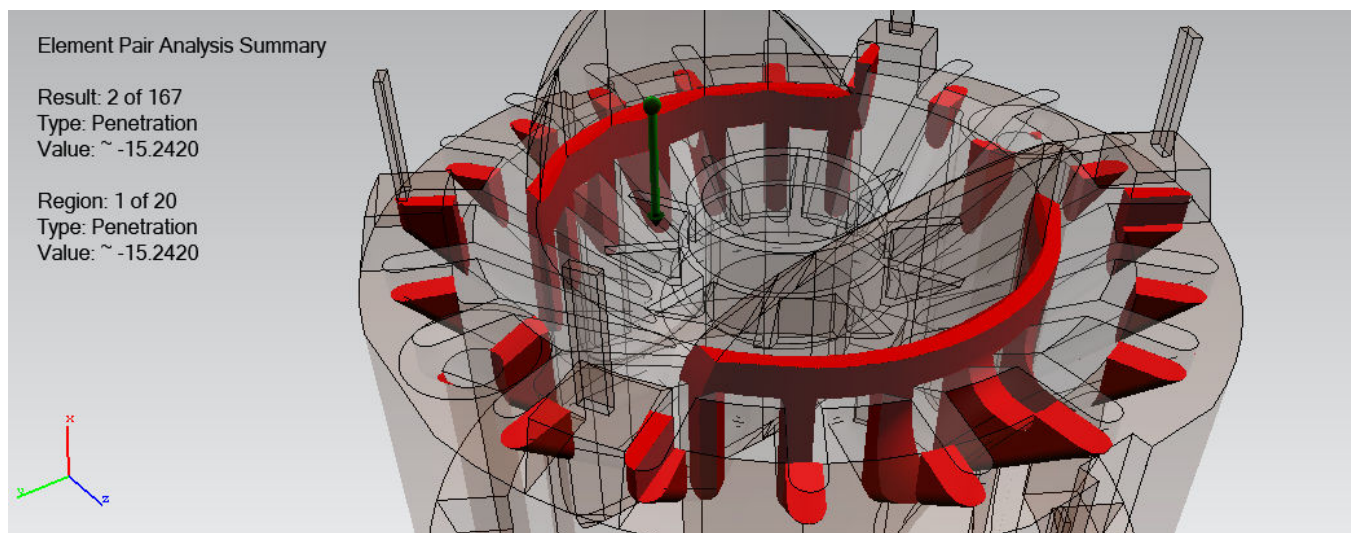
6. On the **Clearance Results** toolbar, click **Analysis Pair Visibility** .
7. In the **Analysis Pair Visibility** dialog box, do one or more of the following.

To	Do this
Change the transparency of one or both of the elements.	a. Select an element checkbox. b. Move the transparency slider to make the element more or less transparent.
Change the transparency of the intersection volume.	a. Select the Intersection Volume checkbox. b. Move the transparency slider to make the intersection more or less transparent.
Turn off annotations.	Clear the Annotations checkbox.

8. Click **OK**.

Display contact and penetration regions

Highlight the precise regions of contact and penetration between element pairs involved in clearance issues.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Window** .
3. Perform a clearance analysis.

The **Clearance Testing** dialog box appears, reporting the progress of the analysis. When finished, an elements list displays in the **Clearance Results** window.

4. On the **3D Clearance** toolbar, click **Analyze Clearance** .

-or-

In the **Clearance Results** window, right-click a clearance issue and choose **Analyze Clearance**.

In the **Clearance Navigation & Analysis** dialog box, all regions of contact and penetration between the element pair are displayed in the **Regions** list. If the element pair has multiple part pairs, and the **Analyze one part pair at a time preference** is activated, regions of contact and penetration between the current analyzed part pair are displayed.

If part pairs are displayed, the **Results** section includes the **Part Pair** navigation, which specifies the analyzed part pair number out of the total number of part pairs in this element pair.



In the Viewing window, the camera zooms to the first, most severe region in the list. Contact and penetration points are displayed in the color red.

Information displayed in the **Regions** list is ordered by the severity of the clearance violation and includes the following:

This column	Contains this information
Region	The region where the contact or penetration occurred. The check box controlling the visibility of contact or penetration points.
Result	The distance between the elements. ~ indicates that the distance is calculated on tessellated data. If you selected Use NURBS in the Clearance Calculator Settings dialog box, the distances of points of

This column	Contains this information
	contact (those with a clearance requirement of 0) are calculated exactly. All other distances are calculated with tessellated data.
Location	The average center and average normal vector of the point of contact between the element pairs.

5. (Optional) If the **Regions** list contains more than one region, move between each **Element Pair** and **Part Pair**, if shown, in the following ways:

Click	For this	
	First Region	Jump to the first region.
	Previous Region	Jump to the previous region.
	Next Region	Jump to the next region.
	Last Region	Jump to the last region.

The Viewing window zooms in on the currently selected region.

Tip:

- **Previous Region** and **Next Region** also are available from the **3D Clearance** toolbar.
- Clear the check box for any region that interferes with your view of another region.

6. (Optional) If the **Clearance Results** window contains more than one issue, move between them in the following ways:

Click	For this	
	First Result	Jump to the first result.
	Previous Result	Jump to the previous result.
	Next Result	Jump to the next result.
	Last Result	Jump to the last result.

The **Regions** list displays each region of the selected result, and the Viewing window displays the first region of contact or penetration.

Tip:

- **Previous Result** and **Next Result** also are available from the **3D Clearance** toolbar.
- Deselect **Analyze** if you do not want to analyze an element pair each time you jump to a new result.

7. (Optional) Use the **Zoom for current region** options to adjust the view of the displayed result in the Viewing window.

Use this option	To do this
+10% (+ key on keyboard)	Zoom in on the displayed result in 10% increments.
-10% (- key on keyboard)	Zoom out on the displayed result in 10% increments.
Apply user camera settings (Ctrl+ + key on keyboard)	Zoom in on the displayed result as defined in the camera settings on the Result page of the Clearance Results Preferences dialog box. Note: When the Only apply when requested preference is not selected, the camera zooms to display 100% percent of the result, and the camera settings are not used.
Fit All (Ctrl+ - key on keyboard)	Fit the displayed result in the Viewing window.

8. The data appears in a new Viewing window. This window is a 3D document, which can be saved and imported into your CAD package to help resolve the clearance issue.
9. (Optional) To rotate your view of the 3D model 180 degrees, click **Flip View**.

The camera rotates to display an opposite view of the selected region.

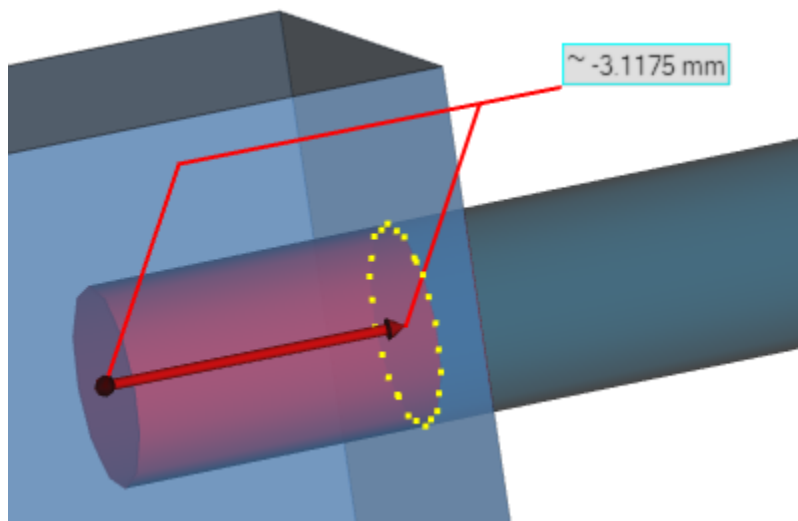
10. (Optional) Select **Show** in the **Distance Lines Study** section if you want to view **distance lines** between two elements that are closer to one another than the specified clearance requirement; then use the **Min** and **Max** sliders to filter the display of distance lines.

Note:

- The maximum filter range is from zero to the value of the specified clearance requirement. Use the sliders or enter values to narrow the filter range. Select **Lock study range** to move both sliders simultaneously. Adjusting the filter automatically updates the number of distance lines displayed in the 3D Viewing window.
- The title of this section changes to **No Distance Lines Study** if the **Create Distance Lines** checkbox is not enabled on the **Clearance Results Preferences** dialog box. In this situation you cannot filter the distance lines.

Display penetration vectors

You can specify to display a penetration vector for element pairs involved in a clearance violation. A penetration vector is an arrow that visually represents the depth of a penetration. The length of the arrow corresponds to the numerical value in the **Result** column of the **Results** list.



A penetration vector

Results				
Item Name	Type	Element Number1	Element Number2	Result
Clearance Results (8/8)				
Result (1/8)	P	Part1	bail_wire2	~ -3.1175
Result (2/8)	P	Part4	bail_wire2	~ -1.8190
Result (3/8)	P	Part5	Part4	~ -0.0615
Result (4/8)	C	handle	handle1_2	~ 0.0000
Result (5/8)	C	drag_knob	body	~ 0.0000
Result (6/8)	C	Part1	rotor	~ 0.0000
Result (7/8)	C	handle1_2	body	~ 0.0000
Result (8/8)	C	Part5	rotor	~ 0.0000

The Results list

Note:

The penetration vector is displayed for penetrations only. It is not displayed for contact or distance issue types.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Window** .
3. Perform a clearance analysis.
4. On the **3D Clearance** toolbar, click **Results Preferences** .

-or-

On the **Clearance Results** toolbar, click **Preferences** .

5. On the **Results** tab of the **Clearance Results Preferences** dialog box, in the **Penetration Vector** section, click the **Show** check box.
6. Click the color well and choose a color for the penetration vector.
7. For **Displayed Regions**, specify how penetration vectors appear in the Viewing window when you double-click or otherwise select penetrations in the **Region** column of the **Clearance Navigation & Analysis** dialog box. Choose from the following options:
 - **All** displays penetration vectors for all penetration point regions.

- **Cumulative** displays penetration vectors for all regions as they are gradually analyzed and selected.
- **Selected** displays penetration vectors for only the currently selected regions.

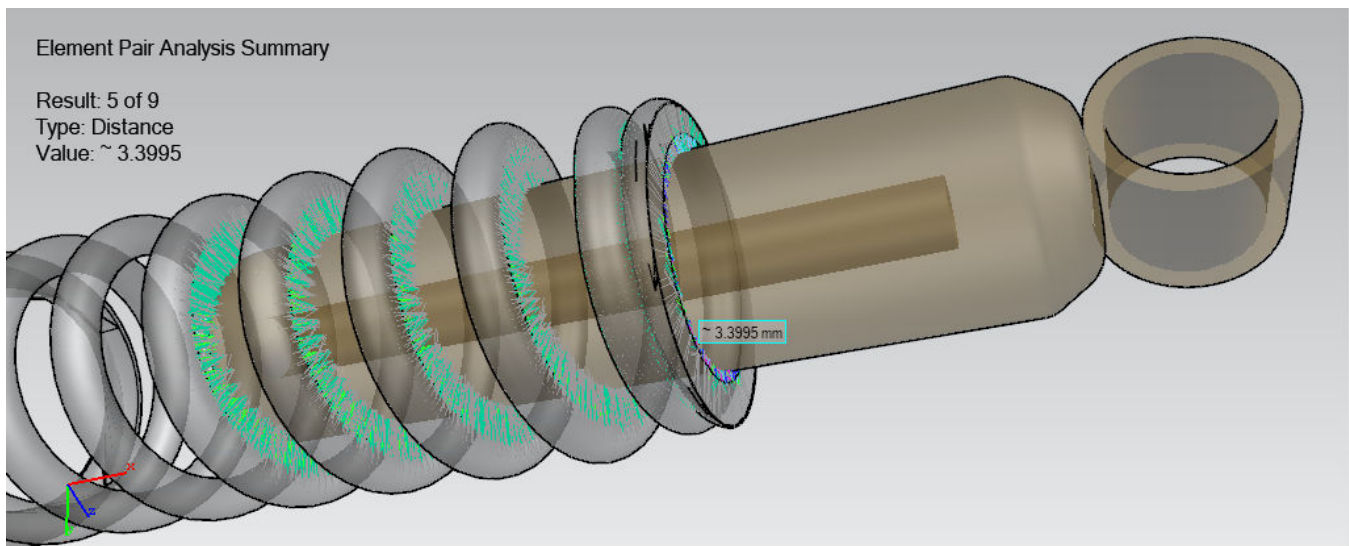
8. Click **OK**.

Display distance lines

Display distance lines between elements that are closer to one another than the specified clearance requirement. Distance lines can be created using up to 10 color ranges and line thicknesses to visually distinguish between results in the Viewing window. Distance lines also can be measured and sent to a new 3D Viewing window.

Note:

- Distance lines are calculated from tessellated geometry only, even for parts that contain B-Rep information.
- The total number of distance lines drawn is based on the tessellation quality of the geometry.
- Distance lines may not be drawn correctly for parts that are contained within other parts.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Window** .
3. Set a **clearance requirement** value greater than zero.

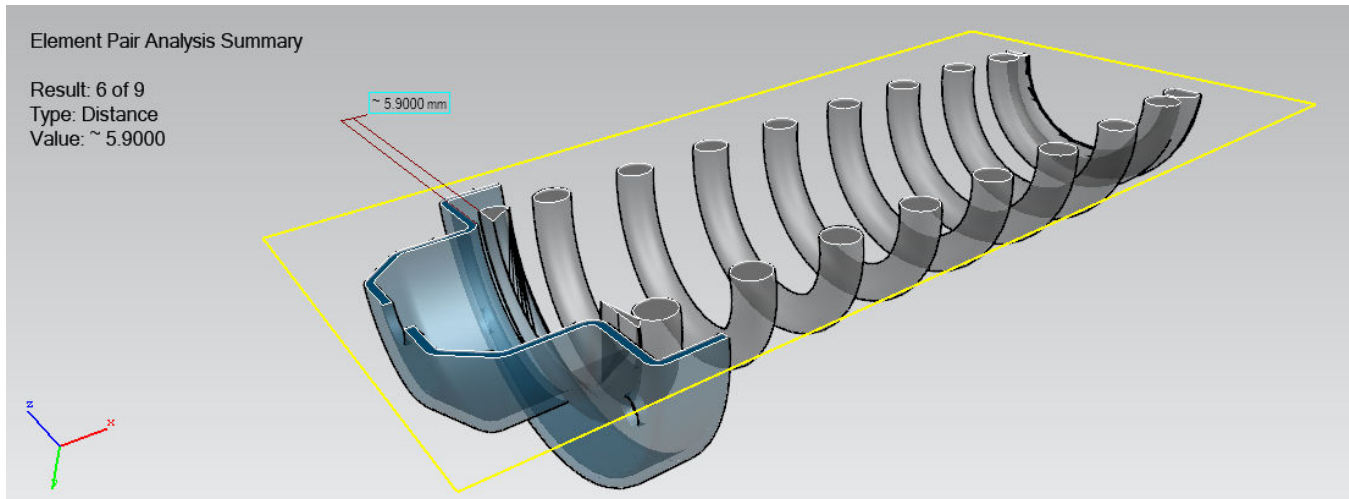
4. Perform a clearance analysis.
5. Do any of the following:

To	Do this
Automatically display distance lines when you double-click a result in the Results list.	a. On the 3D Clearance toolbar, click Results Preferences . b. On the Distance Lines page of the Clearance Results Preferences dialog box, select Create Distance Lines during element pair analysis during element pair analysis. When you double-click a result in the Results list, it is analyzed. If the distance between the element pair equals or is greater than the specified clearance requirement, distance lines are displayed between the elements in the Viewing window.
Create distance lines using the Clearance Navigation & Analysis dialog box.	a. In the Results list, right-click a result and choose Analyze Clearance. b. In the Clearance Navigation & Analysis dialog box, select Show in the No Distance Lines Study section. If the distance between the analyzed element pair equals or is greater than the specified clearance requirement, distance lines are displayed between the elements in the Viewing window. c. (Optional) To filter the display of distance lines, adjust the sliders in the Distance Lines Study section. Note: • The maximum filter range is from zero to the value of the specified clearance requirement. Use the sliders or enter values to narrow the filter range. Select Lock study range to move both sliders simultaneously. Adjusting the filter automatically updates the number of distance lines displayed in the 3D Viewing window. • The title of this section changes to No Distance Lines Study if the Create Distance Lines during element pair analysis checkbox is not enabled on the Clearance Results Preferences dialog box. In this situation you cannot filter the distance lines.
Specify color ranges and line thicknesses.	a. On the 3D Clearance toolbar, click Clearance Results Preferences .

To	Do this
	b. On the Distance Lines page of the Clearance Results Preferences dialog box, choose the number of color ranges you want to create. You can create up to 10. c. For each color range, do the following: • If Absolute ranges are selected, specify an upper limit. Note: • You need to specify only the upper limit for each color range. The application uses this value as the lower limit for the next color range that you create. If the values used for the upper and lower limits of the color ranges have gaps or overlap, when you click OK or Apply the application displays a warning. • Specify a color. • Specify a line thickness. Note: The unit of measurement used for color ranges is the unit of measurement of the currently loaded model.
Measure distance lines.	In the Viewing window, click a distance line. Note: The unit of measurement displayed in the measurement label is the unit of measurement of the currently loaded model.

Display cross sections

Display cross section planes at regions of contact or penetration.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Preferences** .
- or-
- On the **Clearance Results** toolbar, click **Preferences** .
3. In the **Clearance Results Preferences** dialog box, click the **Sections** tab.
4. Select the **Show** check box.
5. In the **Displayed Regions** section, select one of the following options:
 - **All** creates cross sections for all contact and penetration point regions.
 - **Cumulative** creates cross sections for all regions as they are gradually analyzed and selected.
 - **Selected** creates cross sections for only the currently selected regions.
6. In the **Clip** section, select one of the following options:
 - **Near** clips away the near side of the 3D model.
 - **Far** clips away the far side of the 3D model.
 - **Both** clips away the 3D model on both sides of the plane, leaving only the cross section.
 - **None** applies no clipping to the 3D model.

7. If you want to delete existing cross sections associated with clearance issues when another clearance issue is analyzed, select the **Discard after finished** check box.
8. If you do not want to automatically display a cross section plane at the location of a contact or penetration, clear the **Show** check box.
9. Click **OK**.
10. Perform a clearance analysis.

The results of the clearance analysis are displayed in the **Clearance Results** window.

If you selected the **Show** check box in the **Clearance Results Preferences**, a cross section appears each time you analyze a clearance issue. In the **Clearance Results** window, right-click a clearance issue, and select **Analyze Clearance**.

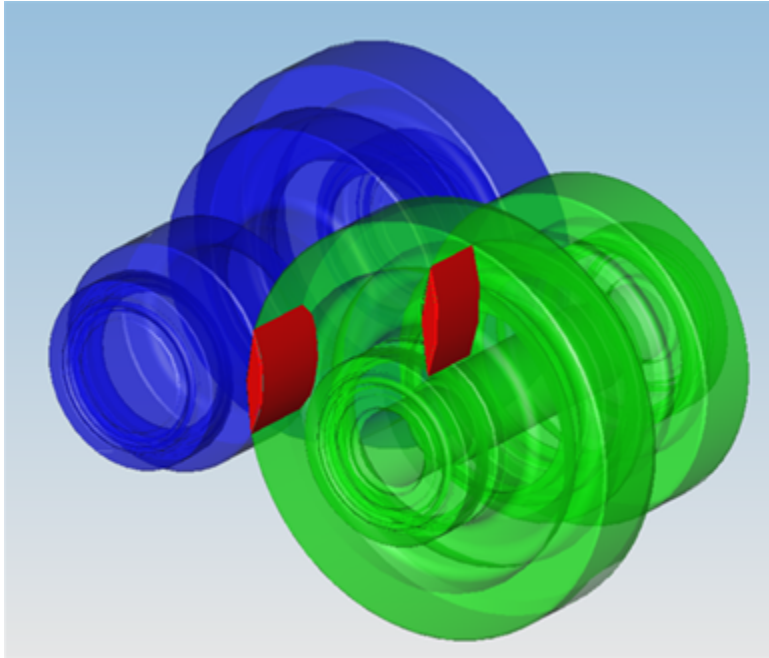
If you want to manually display or remove a cross section plane, right-click a clearance result, and choose **Create Section**.

Display intersection volumes

You can specify to display an intersection volume between any two penetrating elements involved in a clearance violation.

Note:

If **Display Only** intersection volume is enabled on the **Analysis** page, then **Tessellation Lines** and **Feature Edges** cannot be displayed for penetration (P) results.



1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Window** .
3. Perform a clearance analysis.
4. On the **3D Clearance** toolbar, click **Results Preferences** .
5. On the **Analysis** tab of the **Clearance Results Preferences** dialog box, in the **Analysis** section, select the **Analyze** check box.
6. In the **Intersection Volume** section, select the **Show** check box.
7. Choose one of the following:

- **Display only**

Create intersection volumes.

Note:

The **visual quality** of this feature is dependent upon graphics hardware and software. It requires a minimum of OpenGL 3.3 to function properly. To check your current OpenGL version, right-click in the Viewing window, choose **Performance**→**Edit**, and then on the **Troubleshoot** tab, review the **OpenGL** section.

- **Precise (Requires XT B-Rep)**

Create intersection volumes using XT B-Rep data.

8. Click **OK**.

Set clearance results preferences

Adjust the clearance results preferences to change the appearance and behavior of clearance results.

1. On the **3D Clearance** toolbar, click **Clearance** .
2. On the **3D Clearance** toolbar, click **Results Preferences** .
3. In the **Clearance Results Preferences** dialog box, specify any of the following preferences:

On this page	Use this option	To
General	Show annotations in the 3D view when displaying issues	When you select an element pair in the Results list, display the values of the visible columns in an annotation in the Viewing window, along with the elements.
	Load only results associated with elements in the current document	Choose whether to load only the results related to items in the current document or results for the entire product.
	Merge incoming results with	When you create a new list, compare the results against the previous list. List items that appear on the previous list but not the current list are marked as fixed.

On this page	Use this option	To
	existing list	
	Always prompt for saving results before clearing/replacing them	When new results are loaded, prompt to save the current results list.
	Always warn when clearing results manually	When you choose to clear the results list, prompt to confirm or cancel.
	Show results legend	Display clearance type descriptions as a legend within the Clearance Results window.
	Show column filters	Displays or hides the column filter option on the Results list. Type 
	Show 4GD partition scheme legend	Display the partition scheme mapping as a legend within the Clearance Results window.
	Display the Clearance Results and Clearance	Display the Clearance Results and Clearance Graph columns in the Assembly view during clearance analysis. The Clearance Results column shows the number of results for each assembly. The Clearance Graph column displays a bar graph showing the distribution of the different results types for each assembly.

On this page	Use this option	To
	Graph columns in the assembly tree when Clearance is enabled	
	Generate images during clearance analysis	Generate a 2D image of the element pairs involved in each clearance violation. - Select the Generate images during clearance analysis check box. - If you want to confirm image generation each time clearance analysis is performed, select the Show warning dialog check box. - Click Browse to specify a directory for the generated images. - In the Width and Height boxes, type the dimensions to use for the generated images. The default size is 1024x1024. Note: If no value (or 0) is entered in both the Width and Height boxes, the exported image defaults to 1280x1024. If only one dimension value is defined, the image is exported as a square, matching either the width to the specified height or the height to the specified width. - For Image Type, specify the image format. You can choose from the following formats: • JPG • PNG

On this page	Use this option	To
		f. (Optional) Create multiple images per clearance result.
	Zone List Order	Select the order in which you want zones to appear in the Zone list on the Disposition tab of the Clearance Results window. If the list is Unsorted , zones are listed in the order they appear in the zone file.
	Show the Dynamic Clearance Preference Dialog	Select to open the Dynamic Clearance dialog each time a dynamic clearance task is started. This preference overrides the Set as default, do not show this dialog again until requested in Clearance Results Preferences option on the Dynamic Clearance dialog.
Columns	Window	Choose whether you are selecting the columns for the Results or Occurrences window.
	Add >	Add columns of information to the Results list. a. In the Optional Columns list, select the columns to display. b. Click Add > . The options appear in the Visible Columns list. When you generate a list, the new columns appear in the Results list.
	Remove <	Remove columns of information from the Results list. a. In the Visible Columns list, select the columns to remove. b. Click Remove < . The options are removed from the Visible Columns list. When you generate a list, the columns do not appear in the Results list.
	Up Down	Reorder columns of information in the Results list. a. In the Visible Columns list, select the column to move.

On this page	Use this option	To
		b. Click Up to move the column up the list and to the left in the window. c. Click Down to move the column down the list and to the right in the window. The options are reordered in the Visible Columns list. When you generate a list, the columns appear in the new order, from left to right, in the Results list.
	Generate optional 4GD partition scheme columns	Choose whether to generate 4GD partition attribute list for optional columns.
Element Display	Element	Set the distinct display appearance of each element category as it will appear in the Viewing window when you select a clearance result in the Results list.
	Shaded	Apply color and transparency options to the selected Element category.
	Tessellation Lines	Define the appearance of tessellation lines for the selected Element category. If Display Only intersection volume is enabled on the Analysis page, then Tessellation Lines and Feature Edges cannot be displayed for penetration (P) results.
	Feature Edges	Define the appearance of feature edges, used in the parasolid kernel of various CAD/CAM packages, for the selected Element category. If Display Only intersection volume is enabled on the Analysis page, then Tessellation Lines and Feature Edges cannot be displayed for penetration (P) results.
	Set As Default	Apply your settings not only to your current session but to all future sessions.
	Reset	Revert to the default Element Display page settings.
Analysis	Analyze	Automatically perform detailed analysis on an element pair when it is double-clicked in the Results list.

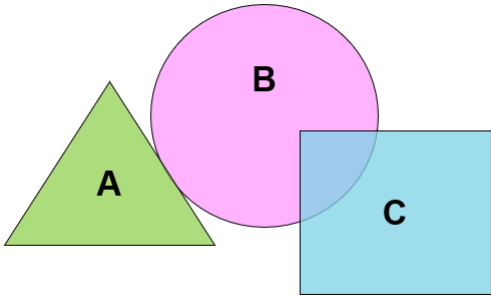
On this page	Use this option	To
	Show dialog	Automatically launch the Clearance Navigation & Analysis dialog box whenever you double-click an element pair in the Results list.
	Analyze one part pair at a time	For elements that have multiple parts, enable the option to analyze individual part pairs within elements one at a time, instead of all part pairs in the elements collectively, in the Clearance Navigation & Analysis dialog box. Note: You must select the Analyze check box to activate the Analyze one part pair at a time option.
	Show Contacts	Automatically generate contact information for each element pair you double-click the Results list. This information appears in the Region column of the Clearance Navigation & Analysis dialog box. Note: You must select the Show dialog check box to activate the Show Contacts option.
	Show Penetrations	Automatically generate penetration information for each element pair you double-click in the Results list. This information appears in the Region column of the Clearance Navigation & Analysis dialog box. Note: You must select the Show dialog check box to activate the Show Penetrations option.
	Show (Intersection Volume section)	Automatically display interference geometry for element pairs that are closer than the specified clearance requirement. Choose from the following types of interference geometry: Note: When you enter element pair analysis mode, Display only is automatically selected. When you leave element analysis mode, the original selection for the Show

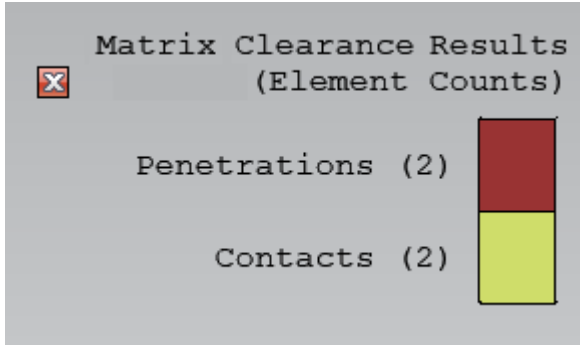
On this page	Use this option	To
		option is enabled. Leaving element pair analysis mode is defined as one of these four actions: • You disable clearance analysis. • You close the Clearance Results window. • You choose Clear Analysis. • You close the open document. • Display only This option uses Advanced Transparency. Note: If the Display only intersection volume option on the Analysis tab is selected, then Tessellation Lines and Feature Edges cannot be displayed for penetration (P) results. • Precise This option requires XT B-Rep data. Note: You must select the Analyze check box to activate the Show option.
	Show (Element Pair Analysis Summary section)	Display the selected element pair results in the Viewing window. • Type or scroll to set the font Text for the display. • Result Type & Value displays the element pair result type and number. • Region Type & Value displays a region of the element pair result.

On this page	Use this option	To
		• Element Names displays the element pair names.
	Show the best-fit planes for each region	Displays the plane angle used to calculate the clearance. The rays used to calculate the penetration depth are shot perpendicular to this plane. When the color section is set to Automatic, the plane color and region color are the same. These values are constrained by the Displayed Region selection. For example, if the Displayed Region is set to Selected, then only the selected regions will show the planes.
	Show the penetration rays for the current region	Provides a graphical representation of each ray that was considered in the penetration depth calculation. The green ray represents the actual ray used to calculate the depth, which is the longest. The gray rays that display next to the green ray were the alternate rays considered in the calculation. These values are constrained by the Displayed Region selection. For example, if the Displayed Region is set to Selected, then only the selected regions will show the rays.
	Show the numbers for each region	Displays the number and type of each penetration region shown in the Clearance Navigation & Analysis dialog. For example, P1 corresponds to the penetration 1 region and C3 corresponds to the contact 3 region. These values are constrained by the Displayed Region selection. For example, if the Displayed Region is set to Selected, then only the selected regions will show the region numbers.
	Show (Contact and Penetration Points section)	Enable all options in the Contact and Penetration Points area of the Analysis page. Note: Show must be selected to automatically display selected Analysis options in the Viewing window, including Create intersection volume, Show contacts, and Show presentations.
	Color Assignment	Display contact and penetration points using default (Automatic) or custom (Manual) colors so that each region type and its severity are easily discernible in the Viewing window. When Automatic is selected, the custom color selectors are disabled.

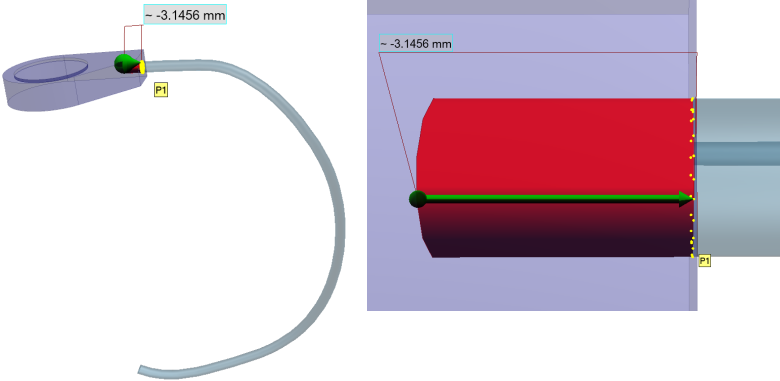
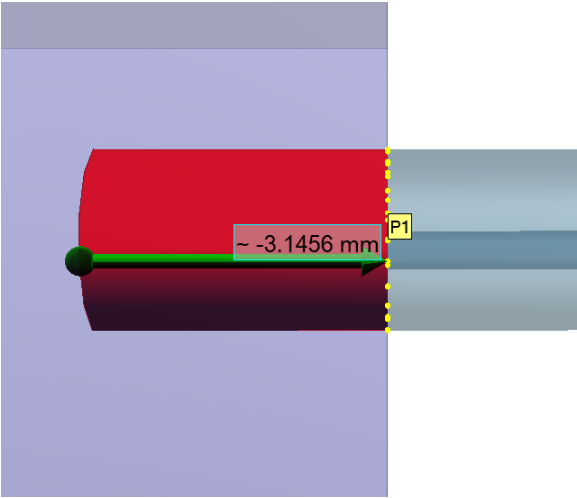
On this page	Use this option	To
		You may select a custom color for any or all of the following types, and change their Size from 1-16 pixels: • All contacts displays all contact points in the selected color and size • Most severe penetrations displays the most extreme penetration points in the selected color and size • Less severe penetrations displays the least extreme penetration points in the selected color and size • Selected contacts and penetrations displays only the most important contact or penetration points selected in the Results list, or those contacts or penetrations selected in the Clearance Navigation & Analysis list
	Display Region s	Indicate how violation regions appear in the Viewing window when you select contacts and penetrations in the Region column of the Clearance Navigation & Analysis dialog box. Select one of the following options: • All displays all contact and penetration point regions • Cumulative displays contact and penetration points cumulatively as regions are selected. • Selected displays only the currently selected regions
Distance Lines	Create distance lines during element pair analysis	Automatically display distance lines between element pairs that are closer than the specified clearance requirement.
	Absolute	Base distance line ranges on the Clearance Requirement value.
	Relative	Base distance line ranges on a percentile of the Clearance Requirement value, making the value independent of the model scale and values.
	Number of	Color code distance lines based upon their length. You can create up to 10 color ranges, each with a unique line color

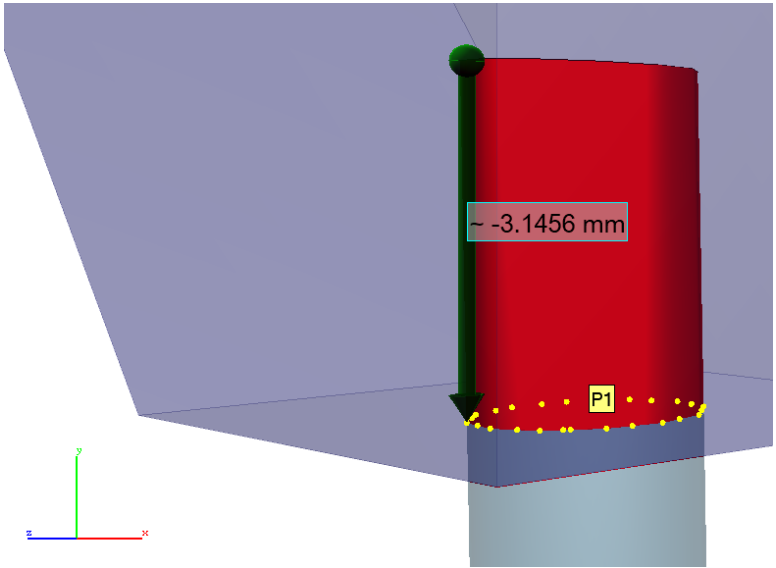
On this page	Use this option	To
	ranges to use	and width. Distance lines that fall within a color range are displayed in the specified color and width.
	Enable (Distance Lines Creation section)	Enable options in the Distance Lines Creation area of the Distance Lines page. - In the first box, enter the filtering interval. - In the second box, enter the maximum number of distance lines. Example: If the first value is 15 and the second value is 5000, the Viewing window draws every 15th line until 5000 lines are displayed.
	Fewer Lines More Lines (Distance Line Density section)	Determine the distance line density using the density slider.
Result	Show (Macro Analysis Summary section)	Display the selected Matrix Clearance Analysis Summary values in the Viewing window. Text Size determines the text size of selected summary values. Time Information displays the Start, End, and Runtime values. Element Information displays the Elements Analyzed and Element Pairs Analyzed values. Result Types displays Penetrations, Contact, Distances, Unknowns, Thicknesses, and Failures values. Analysis Settings displays the Analysis Scope and Element Type values.

On this page	Use this option	To
		• Calculator Settings displays the Requirement and Contact Tolerance values.
	Visual Reports	Specify the content of visual reports in Clearance. • Element displays the number of violating elements. • Results displays the number of violating element pairs. Example:  Results (violating element <i>pairs</i>): • 1 contact (A and B) • 1 penetration (B and C) • 2 total Matrix Clearance Results (Result Counts) Penetrations (1) Contacts (1)  Element B participates in both a contact result and a penetration result. In the visual report, it is displayed in the penetration color, since penetrations are considered a higher priority than contacts. Elements:

On this page	Use this option	To
		• 2 elements in contact (A and B) • 2 elements in penetration (B and C) • 4 total  Element B participates in both a contact result and a penetration result, so it is represented in the counts for both of those groups in the visual report.
	Show (Penetration Vector section)	Display an arrow to visually represent the depth of a clearance penetration. The length of the arrow corresponds to the numerical value in the Result column of the Results list
	Displayed Regions	Indicate how penetration vectors appear as annotations in the Viewing window when you double-click or otherwise select contacts or penetrations in the Region column of the Clearance Navigation & Analysis dialog box. Select one of the following options: • All displays penetration vectors for all contact and penetration point regions • Cumulative displays penetration vectors for all regions as they are gradually analyzed and selected • Selected displays penetration vectors for only the currently selected regions
	Show (Measurement section)	Enable viewing options in the Measurement area of the Result page.

On this page	Use this option	To
	Display ed Regions	Indicate how measurement results appear as annotations in the Viewing window when you double-click or otherwise select contacts or penetrations in the Region column of the Clearance Navigation & Analysis dialog box. Select one of the following options: • All displays measurements for all contact and penetration point regions • Cumulative displays measurements for all regions as they are gradually analyzed and selected • Selected displays measurements for only the currently selected regions
	Show (Bounding Boxes section)	Enable options in the Bounding Boxes area of the Result page.
	Display ed Regions	Indicate how bounding boxes appear in the Viewing window for analysis results when you select contacts or penetrations in the Region column of the Clearance Navigation & Analysis dialog box. Select one of the following options: • All displays bounding boxes for all contact and penetration point regions • Cumulative displays bounding boxes for all regions as they are gradually analyzed and selected • Selected displays bounding boxes for only the currently selected regions
	Zoom to result	In the Viewing window, zoom the display to between 1 and 100 % of the clearance violation area when the element pair is analyzed in the Results list. Zoom to result (0% — "fit all" for element pair) Zoom to result (100% — penetration volume fills the screen)

On this page	Use this option	To
		
	Only apply when requested	When this option is selected, camera settings are not enabled during result navigation, but when you click Apply user camera settings in the Clearance Navigation & Analysis dialog box. This allows the camera to stay zoomed out on the entire element pair when there are multiple regions that need to be investigated.
	Set camera orientation	Automatically orient the camera when a clearance result is analyzed. • Auto-orient — When navigating to an element pair, automatically orient the camera for each contact or penetration region so the penetration vector starts in the middle of the screen and points to the right. This typically provides the best view to see the severity of each contact or penetration region. 

On this page	Use this option	To
		• Align up vector to — When navigating to an element pair, automatically set the camera's "up" vector to the user-defined vector. The camera aligns to the current result or region.  Tip: Press CTRL+ to zoom to the current region.
Sections	Show	When a clearance issue is analyzed, display a cross section plane at the location of a contact or penetration.
	Display ed Regions	Indicate what types of regions of interference are divided into cross sections in the Viewing window when you double-click or otherwise select an element pair in the Results list. Select one of the following options: • All creates cross sections for all contact and penetration point regions. • Cumulative creates cross sections for all regions as they are gradually analyzed and selected. • Selected creates cross sections for only the currently selected regions.

On this page	Use this option	To
	Clip	Indicate how to clip the 3D model to better display the surrounding regions of interference when the cross sections are aligned: • Near clips away the near side of the 3D model. • Far clips away the far side of the 3D model. • Both clips away the 3D model on both sides of the plane, leaving only the cross section. • None applies no clipping to the 3D model.
	Discard after finished	Discard the generated cross sections when a new result is selected in the Results or Clearance Navigation & Analysis list.

4. Click **OK**.

Note:

When you change your preferences, all clearance results lists that you generate appear with the same preferences, except for those on the Element Display page if **Set as Default** is not selected. If you save the results list with additional columns of information, you can change your preferences to display that information. However, if you save the results list with fewer columns of information, those columns are not available.

Saving clearance results to a file or report

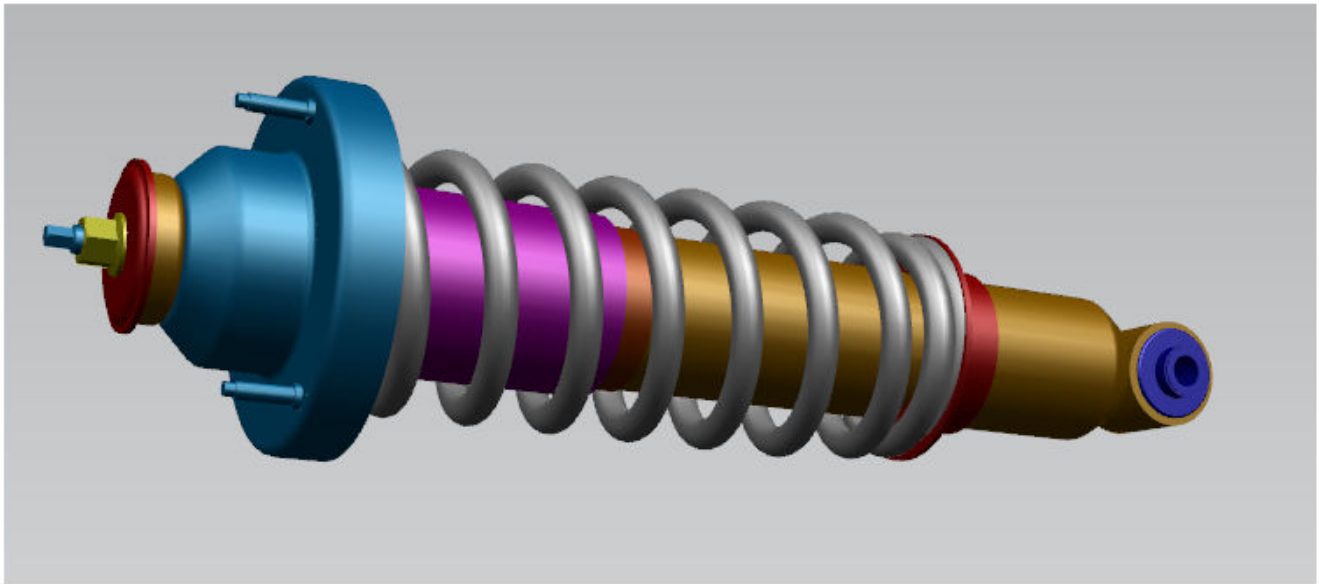
Save clearance results as a report

Once you have generated a clearance results list, you can create a report in a standard or custom format.

Clearance Report - shock_absorber.jt

Table of Contents

- [1.0 Overview](#)
- [2.0 Settings](#)
- [3.0 Summary](#)
- [4.0 Results](#)



1. Perform a clearance analysis.

When the analysis finishes, the list of clearance results is displayed in the **Clearance Results** window.

2. Do one of the following.

- Choose **Clearance**→**Results**→**Report**.
- Right-click an empty area of the **Results** list and select **Results Report**.
- From the **Clearance Results** toolbar, click **Results Report** .

3. In the **Create Report Output** dialog box, **specify your report output and presentation**, and then save your report.

Using a saved list of clearance results

Save a results list as a text file

Note:

References to clearance *results.txt* files are saved when you save your work to a session file. If you have unsaved clearance results, the **Save Clearance Results As** dialog box appears. You must save to a .txt file to reference the results.

References to clearance *results.txt* files are saved when you save your work as PLM XML (if you set the **Save extended 3D content into PLM XML** preference). If you have unsaved clearance results, the **Save Clearance Results As** dialog box appears. You must save to a .txt file to reference the results.

The *results.txt* file is designed for use by Teamcenter lifecycle visualization mockup and is not intended for external use. If you choose to use this file in an external process you do so at your own risk, as this file is unsupported.

Right-click an empty area of the **Results** list, and choose either of the following:

- **Save All**

Save the items that appear in the **Results** list as well as any filtered items.

- **Save Visible**

Save only the items that appear in the **Results** list.

The list of elements and clearance distances is saved as a comma-delimited, quote-enclosed, ASCII text file.

Open a saved results list

1. On the **3D Clearance** toolbar, click **Load Results** .

-or-

On the **Clearance Results** toolbar, click **Load Results** , and then choose **Browse...** or **Quick-load** from the menu.

2. In the **Load Clearance Results** dialog box, type the name of a file or browse to the location of a file, and then click **Open**.

When you check clearance again, the program checks only the elements that have changed since the last clearance check, so the results are available much faster.

Load the last opened results list

Do any of the following to automatically load the last opened results list:

- On the **3D Clearance** toolbar, click **Quick-load results** .
- Right-click an empty area of the **Results** list, and choose **Quick-load**.
- On the **Clearance Results** toolbar, click **Load Results** , and then choose **Quick-load**  from the menu.

Compare current results to a saved results list

1. In the background of the **Results** list, right-click and choose **Compare List to File**.
2. In the **Load** dialog box, browse to the saved results list, select it, and then click **Open**.

The results in the **Results** list are compared with those saved in the elements list. On the line for each result, an icon identifies the state of the results for each element pair.

This icon	Means this
	No difference between the current results and the saved results.
	The location of the contact or interference is different.
	The value of an attribute other than location is different.
No icon	The results for this element pair are not in the saved elements list.

If the values for the **Status**, **Priority**, and **Comment** attributes differ, they are automatically reconciled.

- The value for the **Status** attribute becomes the higher of the two values. Value order (lowest to highest) is **None**, **Low**, **Med**, and **High**.
- The value for the **Priority** attribute becomes the higher of the two values. Value order (lowest to highest) is **Not Evaluated**, **Not Relevant**, **Relevant**, and **Fixed**.
- The value for the **Comment** attribute is changed only if the current results list has no value and the saved elements list has a value. The comment value from the saved elements list appears in the current results list.

The value in the **Number** column depends upon whether or not the element pair exists in the saved results list.

- If the element pair exists in the saved results list, the value in the **Number** column does not change; the element pair retains its original number.
 - If the element pair does not exist in the saved results list, a unique value appears in the **Number** column; the element pair is assigned a number that is one higher than the highest numbered result in the current results list or in the saved results list.
3. (Optional) In the **Results** list, right-click a element pair that has changed, and choose **Reconcile Item**.

Note:

Element pairs that have changed are indicated by **?** in the **Type** column.

The **Reconcile Item** dialog box contains a list of all the attributes and their values for the selected element pair. Attributes whose values have changed are indicated by ***** next to the attribute (column) name.

Columns in the **Reconcile Item** dialog box display the current (after the automatic reconciliation) value, the previous (from the saved elements list file) value, and the pre-comparison (before the automatic reconciliation) value of each attribute.

Managing clearance issues

Overview of managing clearance issues

You can manage your clearance issues by assigning various types of information to individual element pairs in the clearance results list. You can then communicate the clearance issue status with other members of your project team.

Clearance management options include the following:

- **Assign issue disposition** information to an element pair for inclusion in the **Results** list.
- Communicate clearance issues by **email** or by **posting the issues** to Teamcenter Issue Manager.
- **Use zone files** to identify clearance issues by their placement in the model.

Disposition clearance issues

You can specify an issue disposition for each clearance result, capturing such information as owner, priority, status, and more. You can then display the issue disposition information in the clearance results list.

Results							
Item Name	Type	Element Number1	Element Number2	Result	Owner	Priority	
Clearance Results...							
Result (1/24)	P	SPG_SEAT_BLK	UPR_SPG_MT	~ -21.8771	Sophia	Low	
Result (2/24)	P	JNC_BMPR	ROD	~ -21.7809	Olivia	High	
Result (3/24)	P	ROD_BSHG	UPR_BSHG_UPR_SPG_MNT	~ -16.0000	Olivia	High	
Result (4/24)	P	NUT	ROD	~ -15.6000	Olivia	High	
Result (5/24)	P	SHK_BDY	UPR_SHK_BDY_BSHG	~ -12.2885	Ted	Low	
Result (6/24)	P	SPG_SEAT_BLK	STUD	~ -3.1522	Len	Low	
Result (7/24)	P	STUD	UPR_SPG_MT	~ -2.9997	Ted	Low	
Result (8/24)	P	LWR_BSHG_UPR_SPG_MT	ROD_BSHG	~ -1.7678	Olivia	High	
Result (9/24)	P	SPRING	SHK_SLV	~ -0.9375	Sophia	High	
Result (10/24)	P	SPRING	SHK_BDY_SPG_MT	~ -0.5656	Len	Low	

Results list with issue disposition information

1. Perform a clearance analysis.

The results of the clearance analysis are displayed in the **Results** list.

2. If the **Clearance Toolbox** is not already displayed, on the **Clearance Results** toolbar, click **Dispositions** , and then choose **Show/Hide Toolbox** to display the **Clearance Toolbox**.
3. In the **Results** list, select a clearance issue.

Tip:

You can assign information to multiple issues at once by pressing and holding the Ctrl key as you select each issue.

4. In the **Clearance Toolbox**, click the **Disposition** tab.
5. Specify any of the following options:

- **Owner**

Specify the owner of the contact or penetration issue. Click **Add** to type in the names of new owners.

Note:

To use the email feature, you must type the owner in email address format, such as name@yourcompany.com.

- **Priority**

Specify the priority of the contact or penetration issue. Choose from the following:

- **None**
- **Low**
- **Medium**
- **High**
- **Zone**

Specify a zone for the contact or penetration issue. Zones are areas of your model that are defined by you or your company. Click **Add** to add a new zone.

Tip:

You can use the **Zone List Order** preference to set the order in which zones appear in the **Zone** list.

- **Status**

Specify the status of the contact or penetration issue. Choose from the following:

- **Not Evaluated**
- **Not Relevant**
- **Relevant**
- **Fixed**
- **Important**

Flag the issue as important.

- **Change Required**

Flag the issue as requiring a change. If you select the **Change Required** check box, you can also type additional information in the **Change Reference** box.

- **Comment**

Type a comment.

6. Do one of the following:

- **Apply**

Assign the information to the issue.

- **Reset**

Reset the options in the **Clearance Toolbox**.

7. If you want to display issue disposition information in the **Clearance Results** window, do the following:

- a. On the **3D Clearance** toolbar, click **Preferences** .
- b. On the Columns page of the **Clearance Results Preferences** dialog box, in the **Optional Columns** section, select a column to display in the **Clearance Results** window. Choose from the following:
 - **Owner**
 - **Priority**
 - **Zone**
 - **Status**
 - **Important**
 - **Comment**
 - **Change Required**
 - **Change Reference**
- c. Click **OK**.

Post clearance issues to an issues list

You can post clearance issues to a Teamcenter Issue Manager issues list.

1. Perform a clearance analysis.
2. In the clearance results list, do the following:
 - a. In the clearance results list, select one or more clearance results. For multiple selections, hold Ctrl as you click each issue.

- b. Right-click the clearance result, and select **Create Issue(s)**.

Communicate clearance issues by email

1. Perform a clearance analysis.
2. Choose **Clearance**→**Preferences**→**Email**.
3. In the **Clearance Email Preferences** dialog box, select the **Email the owner(s) of the clearance issue** check box.
4. Click **OK**.
5. In the **Results** list, select an issue.
6. On the **Clearance Results** toolbar, click **Dispositions** , and then choose **Show/Hide Toolbox** to display the **Clearance Toolbox**.
7. In the **Clearance Toolbox**, click the **Disposition** tab.
8. For **Owner**, click **Add**.
9. In the **Add Owner** box, type the email address of the issue owner.

Example:

name@yourcompany.com

10. In the **Results** list, right-click the issue and choose **Email Owners**.

An email message is sent to the owner of the issue. The message includes information about the issue, including the analysis results and metadata associated with the element pair.

Note:

To send email from the viewer through a MAPI email program like Microsoft Outlook, the MAPI program must be 64-bit. If it is not, you get a warning message and no message is sent.

Create Teamcenter visual issues from clearance results

You can create Teamcenter visual issues from clearance results. A separate issue is created for each specified result.

1. From Teamcenter, open a 3D model in the Lifecycle Viewer or the standalone viewer.

2. Perform a clearance check.
3. In the **Results** list, select one or more clearance results. For multiple selections, hold Ctrl as you click each issue.
4. Right-click the clearance result, and select **Create Issue(s)**.

The **Visual Issue** dialog box appears.

5. From the **Type** list, choose **Issue report**.

Note:

Issue Report is the default issue type. Your organization may also allow you to choose from additional issue types.

6. In the **Name** box, type a name for the issue report.
7. In the **Data Type for state information** section, choose how you want to capture the state of the viewer:

- **Session File**
- **Product View or 2D Snapshot**

Depending upon your data and your settings in the Teamcenter Integration Visual Issue Preferences, the viewer may generate a 2D snapshot or product view when the issue is created. Choose one of the following Teamcenter relation types for the dataset:

- **Snapshot Before Fix**
- **Snapshot After Fix**

8. Click **OK**.

The **Issue Attribute Dialog** appears, displaying the available attributes.

9. In the **Issue Attribute Dialog**, do the following:
 - a. Double-click **Synopsis** to provide a name for the issue report revision.

The **Attribute Edit** dialog box appears.

- b. In **Value**, type a name for the issue report revision.
 - c. Double-click any of the other attributes to add information relevant to the issue.

- d. Click **OK**.

The **Save As** dialog box appears.

- e. Type a name for the clearance results file, and click **Save**.

The clearance results file is saved to the database.

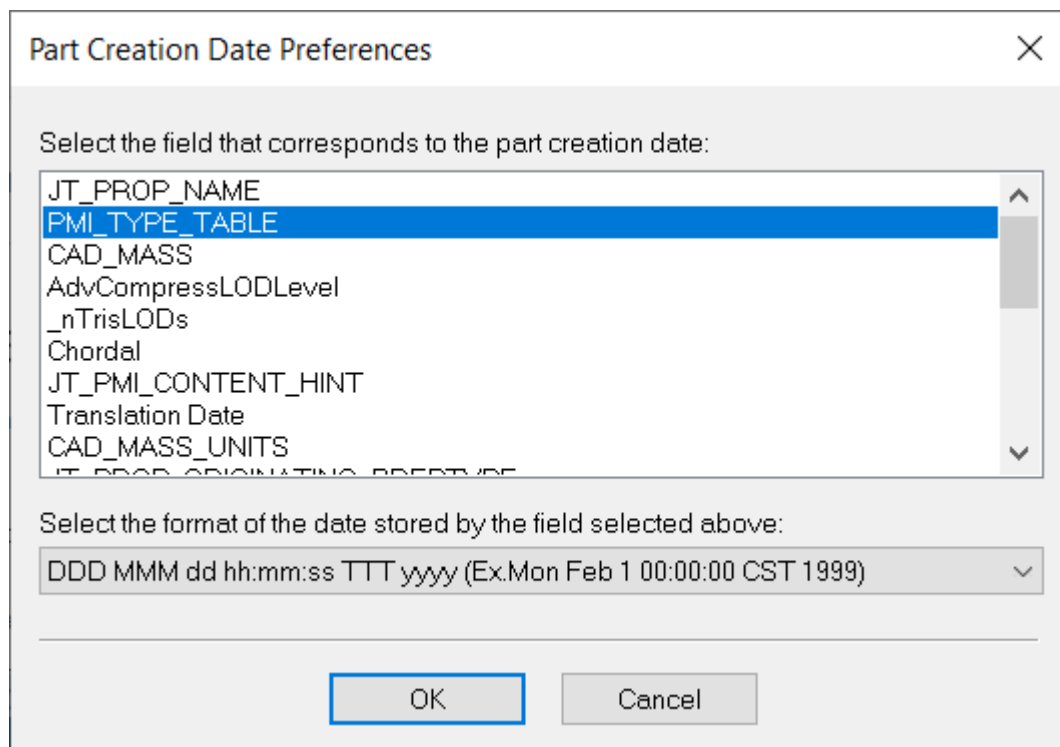
- f. If you are posting multiple clearance results, the **Visual Issue** and **Issue Attribute Dialog** dialog boxes are displayed again. Complete the steps described above for each issue.

A new visual issue is created for each clearance issue selected in the **Results** list.

Set Clearance Manager preferences

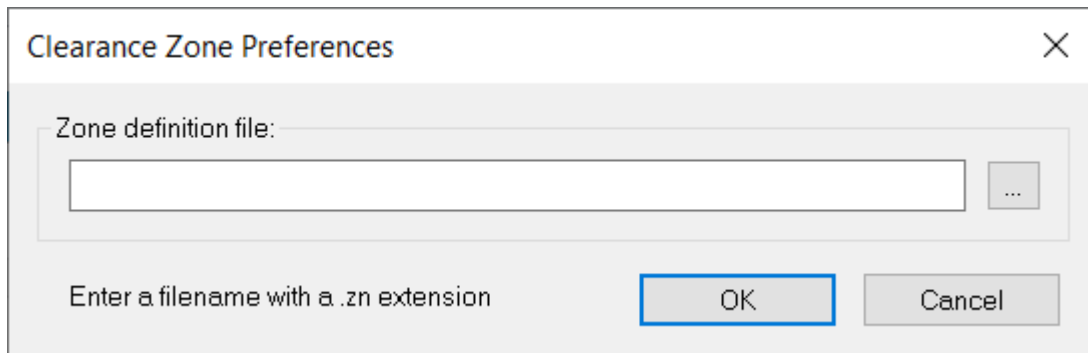
Specify how you want to communicate clearance issues to the appropriate people. You can communicate clearance issues by sending email messages or by posting the issues to an issues list.

1. Identify the attribute that corresponds to the creation date of parts and the format of the date.
 - a. Choose **Clearance**→**Preferences**→**Part Creation Date**.



- b. From the list in the upper portion of the **Part Creation Date Preferences** dialog box, choose the attribute that contains the date a part was created.

- c. From the list in the box in the lower portion of the **Part Creation Date Preferences** dialog box, choose the format by which the date is stored in the selected attribute.
 - d. Click **OK**.
2. (Optional) Identify a list of physical zones for your model.
 - a. Choose **Clearance→Preferences→Zones**.



- b. In **Zone definition file**, type the path to the .zn file that defines your clearance zones, or click the browse button and browse to the file.

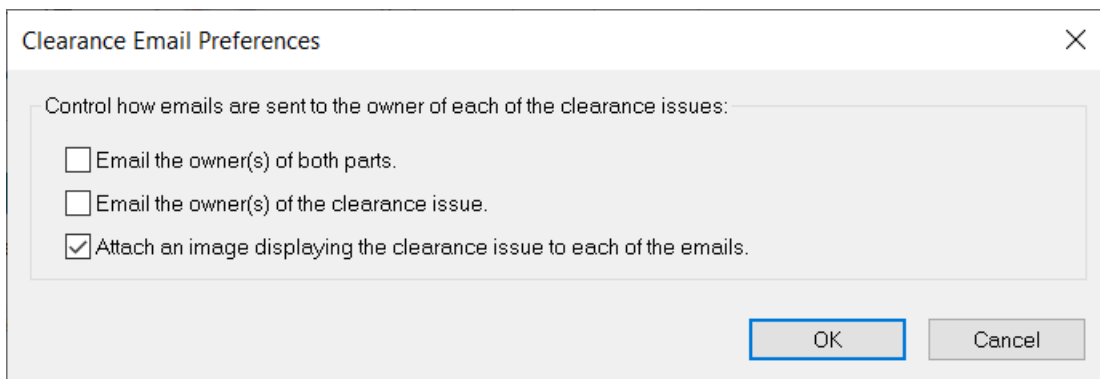
Note:

A .zn file is a tab-separated ASCII file that identifies the zones of your model. You can create your own .zn files.

- c. Click **OK**.

You can include a **Zone** column in the **Results** list, and use the zones to identify the clearance issues that occur in each physical zone of your model.

3. (Optional) To set preferences for sending email messages, choose **Clearance→Preferences→Email**.



4. (If you clicked **Email** in Step 3) In the **Clearance Email Preferences** dialog box, change the preferences in one or more of these ways:

Use this option	To do this
Email the owner(s) of both parts	Send email to the owners of both parts involved in a clearance issue.
Email the owner(s) of the clearance issue	Send email to the owners of the clearance issue.
Attach an image displaying the clearance issue to each of the emails	When you send email messages about clearance issues, attach an image of the part pair to the message.

5. Click **OK**.

When you check clearance or work with the list in the **Clearance Results** window, email messages are sent as you have specified.

Create a zone file

You can create zone files to identify the clearance issues in your model by their placement in the model.

- In a text editor, create a file with the following parameters:
 - The first line can include any information and has no requirements. Its purpose is to make the file readable by humans.
 - Each subsequent line, which defines one zone, must be a tab-separated list of seven values.
 - The first field in each line identifies the name of the zone, as it will appear in the **Results** list.
 - The next six fields identify the minimum and maximum coordinates of a box that completely contains the zone.

Example:

Zone Name	X1	Y1	Z1	X2	Y2	Z2
cockpit	-10.0	-40.0	-60.0	10.0	90.0	90.0
tail	10.0	-40.0	-60.0	30.0	90.0	90.0
starboard wing	30.0	-40.0	-60.0	50.0	90.0	90.0
port wing	50.0	-40.0	-60.0	70.0	90.0	90.0

cargo bay	70.0	-40.0	-60.0	90.0	90.0	90.0
lavatory	90.0	-40.0	-60.0	110.0	90.0	90.0
passenger seating area	110.0	-40.0	-60.0	130.0	90.0	90.0

2. Save the file with a ZN extension.

Specify the zone file when you set Clearance Manager preferences.

View the zones defined by a zone file

1. Choose **Actions**→**View Zones**.
2. In the **Load Zone File** dialog box, open the ZN file.

In the Viewing window, a 3D representation of the zones appears on your model.

Working with variants

Understanding variant-based clearance analysis

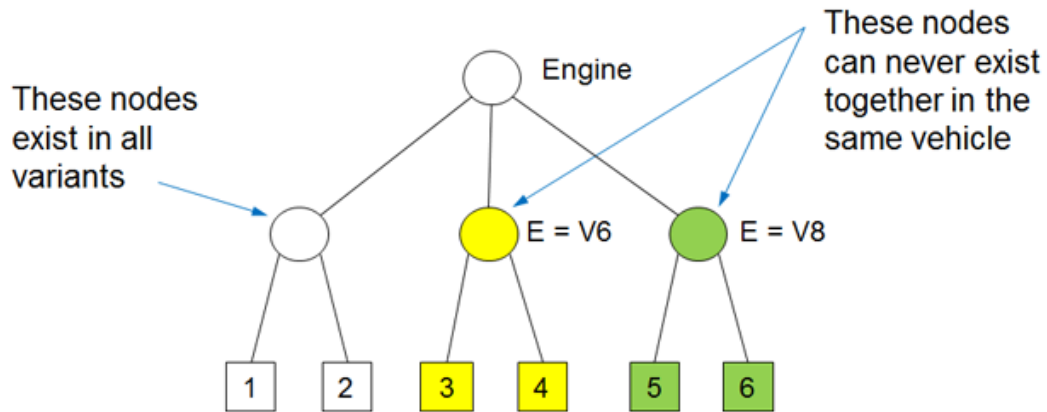
Variant-based clearance analysis uses product variant information stored in Teamcenter. Analyses can be performed on a 150% BOM structure, which is the union of two or more product variants. Non-buildable combinations of product structure are detected and excluded from the clearance analysis.

The benefits of removing non-buildable occurrence relationships from the ClearanceDB analysis include the following:

- Eliminate results between non-buildable product configurations.
- Faster clearance analysis.
- Fewer clearance incidents for engineers and designers to evaluate.

A simple example with variant conditions (150% BOM with variants)

Using variant analysis, a single 150% BOM can contain all buildable product combinations. This enables you to perform a single clearance analysis, which excludes all non-buildable part pairs.



With no variants, all nodes are analyzed (15 part pairs).

No nodes are excluded.

	1	2	3	4	5	6
1						
2	X					
3	X	X				
4	X	X	X			
5	X	X	X	X		
6	X	X	X	X	X	

With the E=V6 variant, nodes 1, 2, 3, and 4 are analyzed (6 part pairs).

Nodes 5 and 6 are excluded (9 part pairs).

	1	2	3	4	5	6
1						
2	X					
3	X	X				
4	X	X	X			
5						
6						

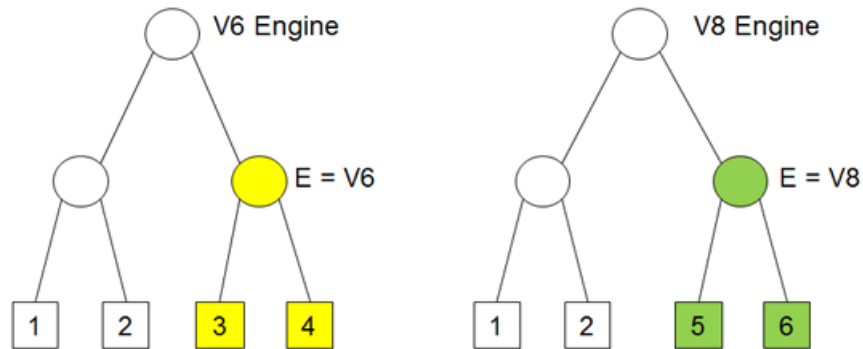
With the E=V8 variant, nodes 1, 2, 5, and 6 are analyzed (6 part pairs).

Nodes 3 and 4 are excluded (9 part pairs).

	1	2	3	4	5	6
1						
2	X					
3						
4						
5	X	X				
6	X	X			X	

A simple example without variant conditions (separate 100% BOMs)

Without using variant analysis, you must create and manage separate 100% BOMs to represent each buildable product combination. You must evaluate each BOM with a separate clearance analysis.



For the analysis of the V6 Engine, without variant conditions, nodes 1, 2, 3, and 4 are analyzed (6 part pairs).

No nodes are excluded.

	1	2	3	4
1				
2	X			
3	X	X		
4	X	X	X	

For the analysis of the V8 Engine, without variant conditions, nodes 1, 2, 5, and 6 are analyzed (6 part pairs).

No nodes are excluded.

	1	2	5	6
1				
2	X			
5	X	X		
6	X	X	X	

For more information about working with Teamcenter variant conditions, see [Structure Management on Rich Client — Usage](#) in the Teamcenter help.

Variant-based clearance analysis methods

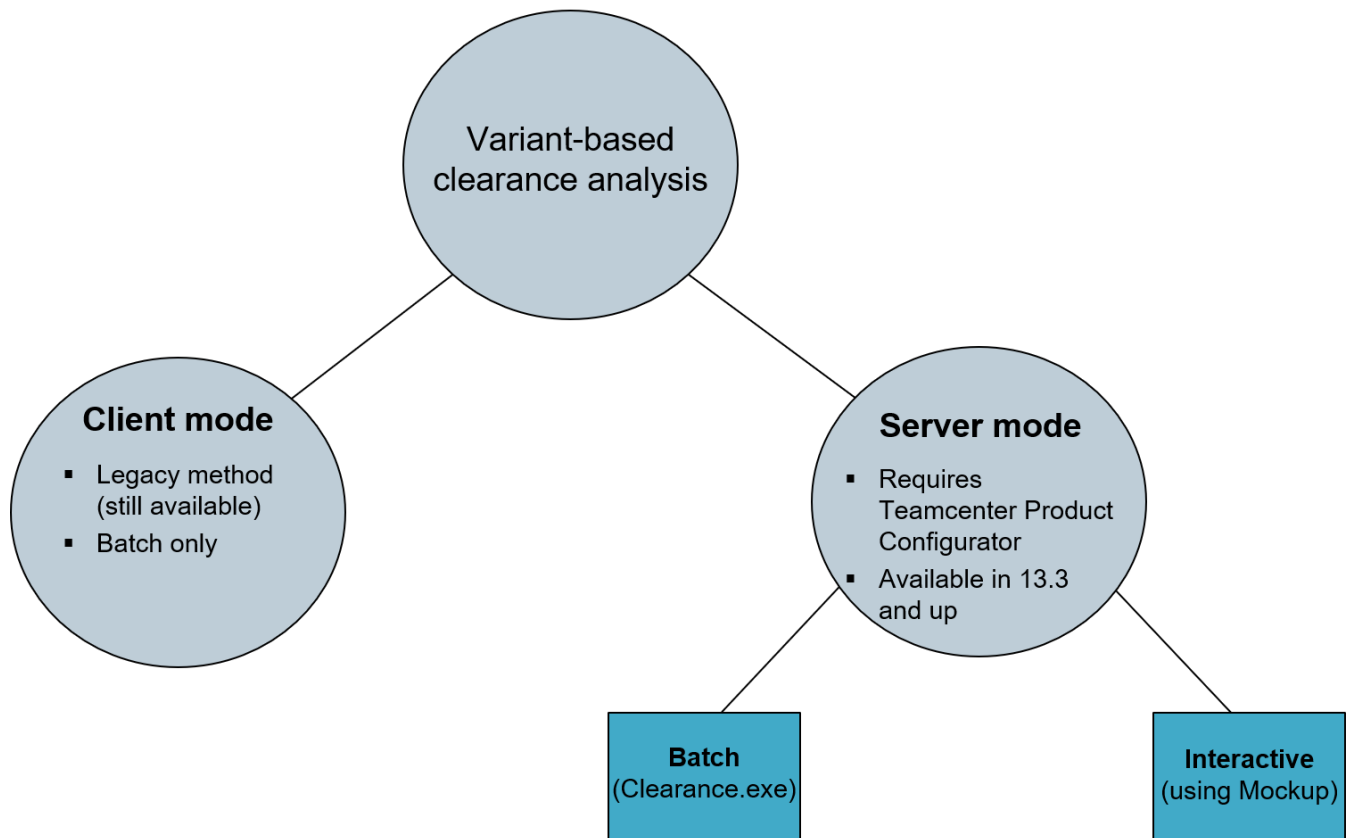
You can perform variant-based clearance analysis using the following methods:

- Client-side
 - No Teamcenter connection is required during the analysis.
 - Can only be run in batch using *Clearance.exe*.
- Server-side

- Requires a connection to Teamcenter Product Configurator during the analysis.
- Can be run either interactively in Teamcenter lifecycle visualization or in batch using *Clearance.exe*.

See Product Configurator on Rich Client to learn more about creating, modifying, and managing variants in Teamcenter.

Teamcenter lifecycle visualization



Variant-based clearance analysis (client-side)

You must enable the option to evaluate ClearanceDB products according to Teamcenter variant conditions.

1. Using a text editor, open *Clearance.cfgglobal* and adjust the following settings:

Note:

You can also specify these options in the *Clearance.cfgproduct* file for the product.

LexicographicalVariantAnalysis

Defines the mode of the evaluation of variant condition relational expressions. Choose one of the following:

No — If indeterminable variant conditions exist, the affected variant inferences are asked from the server. This is the default.

Yes — If indeterminable variant conditions exist, evaluate them lexicographically.

Note:

Variant conditions that cannot be evaluated based on textual representations are indeterminable.

VariantAnalysisClientTraceFile

Specifies the name for the log file of the variant analysis. By default, this is not set, and a log file is not generated.

Example:

VariantAnalysisClientTraceFile=cldb_variant_analysis.log

VariantAnalysisClientTraceFlags

Specifies the content of the log file of the variant analysis. The trace flags are additive, concatenated with the plus (+) character.

Example:

VariantAnalysisClientTraceFlags=CONFIGURATION+STATISTICS

Add any of the following flags:

CONFIGURATION	Lists the variant analysis configuration attributes.
STATISTICS	Provides the basic variant analysis statistics.
VARIANTCONDITIONS	Provides details of textual variant conditions.
DETERMINABILITY	Lists the determinability of variant conditions.
SATISFIABILITY	Lists the satisfiability of variant conditions.
EXCLUDEDVCPAIRS	Lists excluded variant condition pairs.
EXCLUDEDUIDPAIRS	Lists excluded clearance element pairs as ABSOCs.
EXCLUDEDNGIDPAIRS	Lists excluded clearance element pairs as NGIDs.
IMPACT	Lists the impact of variant conditions on the product structure.

Caution:

Enabling the **EXCLUDEDUIDPAIRS** and **EXCLUDEDNGIDPAIRS** trace flags may result in extremely large log files.

VariantLogicalExpression

Define the variant condition UserValue title and BOM line property pair in the .plmxml file. This has the form of:

"Variant Condition":bl_variant_condition

"Variant Formula":bl_formula

Example:

VariantLogicalExpression="VC-V6:bl_variant_condition"

Note:

This must match the content of the **BomWriterUserAttributes** setting.

BomWriterUserAttributes

Specify the variant conditions for the BOMwriter to include in the generated .plmxml file. Type these according to the following syntax:

target:Instance,key:myAttribute,literal:"My Attribute Value"

Note:

This is the same option as the **bomwriter -ua=** command line argument.

Example:

Use this option to include variant model related BOM line properties in the generated .plmxml file. For example:

BomWriterUserAttributes=target:Instance,key: VC-V6,prop: bl_variant_condition

RulesObject = Variant Analysis

Specify to perform analysis using Teamcenter variant conditions. To enable this option, remove the number sign symbol (#) that precedes the **RulesObject = Variant Analysis** line.

2. Save the file.

Variant-based clearance analysis (server-side)**Performing variant-based clearance analysis in Lifecycle Visualization**

1. With an unconfigured .plmxml file open, choose **Menu→Clearance→Preferences→Product Configurator**.

2. In the **Product Configuration** dialog box, do the following.
 - a. Select the **Enable Product Configurator** check box.
 - b. From the **Variant Rule** dropdown, select the desired variant rule or criteria to apply against the loaded file during clearance analysis.
 - c. Click **OK**.
3. Run a clearance analysis.

Performing variant-based clearance analysis in batch

To perform variant analysis in batch mode (*Clearance.exe*) using Teamcenter Product Configurator, complete the following steps.

1. Ensure that you are connected to Teamcenter and that the connection is maintained for the duration of the analysis process.
2. Using a text editor, open *Clearance.cfgglobal* and adjust the following settings:

Note:

You can also specify these options in the *Clearance.cfgproduct* file for the product, which will override settings in the *Clearance.cfgglobal* file.

EnableProductConfigurator

Enables *Clearance.exe* to use the Teamcenter Product Configurator to perform variant analysis.

Example:

EnableProductConfigurator=True

ProductConfiguratorVariantRule

Specifies the saved variant rule that is used for communicating with the Product Configurator.

Example:

ProductConfiguratorVariantRule=<valid variant rule>

VariantLogicalExpression

Define the variant condition UserValue title and BOM line property pair in the .plmxml file. This has the form of:

"Variant Condition":bl_variant_condition

"Variant Formula":bl_formula

Example:

VariantLogicalExpression="VC-V6:bl_variant_condition"

Note:

This must match the content of the **BomWriterUserAttributes** setting.

BomWriterUserAttributes

Specify the variant conditions for the BOMwriter to include in the generated .plmxml file. Type these according to the following syntax:

target:Instance,key:myAttribute,literal:"My Attribute Value"

Note:

This is the same option as the **bomwriter -ua=** command line argument.

Example:

Use this option to include variant model related BOM line properties in the generated .plmxml file. For example:

BomWriterUserAttributes=target:Instance,key: VC-V6,prop: bl_variant_condition

3. Save the file.

Working with ClearanceDB

Work with ClearanceDB

An extension of the Teamcenter lifecycle visualization clearance management functionality, ClearanceDB enables you to persist and manage clearance issues in an Oracle relational database management system.

A clearance issue is a violation of a distance requirement for a pair of element instances. This violation can be

- A minimum distance requirement
- A non-interference requirement

ClearanceDB issues may contain other types of data as well, including

- Type of violation: distance, contact, or penetration
- Owner: who owns each element
- Priority: high, medium, low, none
- Status: user-specified values
- Zone: the geometric region of the violation
- Comment: a user-defined comment

You can define clearance requirements and conditions based on the metadata of the structure or elements. A clearance requirement defines a property (a distance requirement) or a state (exclusion from analysis) for a subset of model data. A clearance requirement condition defines the scope of data to which the requirement applies. Any metadata stored with elements or assemblies can be used to define the conditions for a requirement, and requirements can be applied to a single set of data or a product of two sets of data.

Using ClearanceDB, you can

- Enable ClearanceDB functionality and save your settings in the registry.
- Perform ClearanceDB analysis in real-time within the Viewer.
- Manage clearance issues within an Oracle database.
- View reports of clearance issues.
- Apply color to elements for a visual representation of the state of the clearance issue management process.
- Create Teamcenter visual issues from ClearanceDB results.

Note:

For detailed information on installing and configuring ClearanceDB, see ClearanceDB — Administration.

Managing ClearanceDB issues in the database

Manage ClearanceDB issues in the database

Using ClearanceDB you can load clearance results from the database, evaluate and set the status of any issues, and then update the database with your changes.

Load results from the Clearance database

Tip:

To check the Clearance database for possible updates to the attribute information displayed in the **Clearance Results** window, right-click an element pair and choose **Refresh**.

1. Open a 3D model that has clearance results associated with it in the database.
2. On the **3D Clearance** toolbar, click **Load Results** .

-or-

On the **Clearance Results** toolbar, click **Load Results** , and then choose **Browse...** or **Quick-load** from the menu.

3. In the **Load Clearance Results** dialog box, from the **Files of Type** list, choose **ClearanceDB DataBase Connection (*.dbc)**.
4. Select a DataBase Connection (DBC) file, and click **Open**.

The 3D model and the database content are matched by the name of the top-level assembly node. Results are displayed in the **Results** list.

Update the status of clearance issues in the database

Tip:

To check the ClearanceDB database for possible updates to the attribute information displayed in the **Clearance Results** window, right-click an element pair and choose **Refresh**.

1. Ensure that ClearanceDB is properly enabled.
2. Load the clearance results from the database.
3. If the **Clearance Toolbox** is not already displayed, on the **Clearance Results** toolbar, click **Dispositions** , and then choose **Show/Hide Toolbox** to display the **Clearance Toolbox**.

4. **Assign information to an element pair** by selecting one of these options:

- **Set Owner**
- **Set Status**
- **Set Priority**
- **Set Zone**
- **Enter Resolution**

If your ClearanceDB configuration is set to Autocommit mode, database updates are triggered by changes to any of the above values.

5. (If not in Autocommit mode) Right-click the **Results** list and choose **Commit**.

Note:

The ability to commit local clearance issue modifications to the database is controlled by the product database configuration, and possibly the content of the clearance issue Owner.

Viewing ClearanceDB reports

View ClearanceDB reports

You can view ClearanceDB results as reports that reflect the state of the Clearance database. ClearanceDB reports pertain to product and system statistics, and are classified based on the scope and type of data within a particular report.

Tip:

Use the Part Statistics report to apply color to parts in the Viewing window.

View ClearanceDB results using reports

1. Ensure that ClearanceDB is enabled.
2. Open a 3D model that has ClearanceDB data associated with it in the database.
3. **Load** ClearanceDB data from the database.
4. Right-click an empty area of the **Results** list and choose **ClearanceDB Reports**.

The **ClearanceDB Report** dialog box appears.

5. In the **Report** section of the **ClearanceDB Report** dialog box, choose a report type.
6. (Optional) To define the report scope, in the **Filter** section, select a filter from the drop-down menu.
7. (Optional) To select a time period filter, in the **Time Period** section, select an entry from the drop-down menu.

Tip:

To load or refresh the data displayed in the report, click **Update**.

Save a report

1. Click **Save**.
2. Type a name, and then click **Save**.

The report is saved to a CSV (comma-delimited) file.

Apply color to parts with clearance issues

You can apply color settings to parts in the Viewing window for a visual representation of the state of the clearance issue management process.

Note:

In the Viewing window, parts are identified by color based on their **Status**, **Priority**, and **Db Modified On** attributes. You can control the order of attribute values as they appear within the report, and the order you specify is saved.

For parts associated with a large number of clearance issues, you can order the attribute values so that a single value is assigned to each part. The first attribute value of the clearance issues associated with a part is assigned to the part.

1. Ensure that ClearanceDB is enabled.
2. Open a 3D model that has ClearanceDB data associated with it in the database.
3. **Load** ClearanceDB data from the database.
4. Right-click an empty area of the **Results** list and choose **ClearanceDB Reports**.
5. In the **Report** section of the **ClearanceDB Report** dialog box, select **Element Statistics** from the drop-down menu.
6. Click **Update**.

The data is loaded into the report.

7. In the **ClearanceDB Report** dialog box, click **Apply Color**.
8. In the **Apply Color** dialog box, from the **Issue Attribute** drop-down list, select an attribute.

Values that reflect the content of the report are displayed in the **Attribute Order** section.

9. (Optional) In the **Attribute Order** section, select an attribute, and then click **Up** or **Down** to change the order of an attribute in the report.
10. Click **Attribute Color**.

The color well appears.

11. Select a color for the attribute.
12. In the **Apply To** section, specify how to apply the color settings to the parts in the Viewing window. Choose from the following settings:
 - Selected Parts
 - Visible Parts
 - All Parts
13. Click **Apply**.

The color settings are applied to the parts in the Viewing window.

Issue inheritance

More than one instance of the same violating part pairs may exist in your clearance results. Inheritance relationships can be defined in your database so that these issues are not always duplicated but instead are managed based on their level of similarity. Membership in a specific inheritance category determines to what degree they share clearance issue attributes and dispositions. For example, if an issue meets the criteria for being **Identical**, the issue is shared between occurrences and appears only once in the clearance results. If the issue is later updated or changed, all part occurrences reference the same, updated version of the issue. Also, if these part pair occurrences exist in different products, they can share the same issue. The attributes of **Similar** and **Resembling** issues are shared to a lesser degree and are handled differently from **Identical** issues, as explained below.

1. In the **Results** list, review the issue occurrences.

The **Db Match**, **Db Base**, **Db Issue**, **Db Issue Count** columns provide information about the occurrences.

Three different levels of similarity may be defined for your database, and they appear in the results lists with different symbols, as follows:

Note:

Unrelated issues appear with the  symbol.

- **Identical**

Identical issues are linked when the inheritance process is executed in the database. The master issue (the one that the others are linked to) is the one that was most recently changed.

- **Similar**

Similar issues share a common baseline issue, which is the one that was most recently changed. When the baseline issue is dispositioned, the information in the baseline issue is copied to the other issues in the same group. This copy operation happens only once, and future changes affect only that issue.

- **Resembling**

Same as similar, except the issues are marked as not evaluated in the database. This typically means that you must investigate and evaluate the issue.

2. To group issues by element pair, right-click a column heading, and choose **Group Tree by Elements**.
3. On the **Clearance Results** toolbar, click **Dispositions** .

A window appears below the **Clearance Results** window, providing details about the selected occurrence.

4. Select other issues in the **Results** list to view the details for the occurrences.

5. Transforming parts

Overview of transforming parts

Transforming parts is to translate (move), rotate, and scale parts of a 3D model. You have several transformation tools. Each tool works in a slightly different way.

You can:

- Translate, rotate, and scale parts by using the **Part Transformation** dialog box.
- Translate and rotate parts by using part **manipulators**.
- Translate and rotate parts to reposition them relative to other parts by using the **Relocate** tool.
- Translate and rotate parts to align them with other parts by using **Alignment**.
- Translate parts by dragging them in the Viewing window using **Part Manipulation Mode**.
- Translate a set of parts along an axis by specifying the distance between parts using **Exploded View**.

Note:

- When you **create a snapshot** of your model with part transformations, the transformations are saved with the snapshot. You can return a model to the transformations saved in any snapshot.
- When you save a session, transformations are saved with the session. When you save a session with multiple Viewing windows, all views in all Viewing windows are saved as well.
- If you have your model open in multiple Viewing windows, your transform affects the model view in all the Viewing windows. You cannot have a view of your model completely assembled in one Viewing window, and an exploded view of your model in a second Viewing window.

Transforming parts using the Part Transformation dialog box

Overview of transforming parts using the Part Transformation dialog box

The **Part Transformation** dialog box is one of several tools you can use to transform parts. Using the **Part Transformation** dialog box, you can:

- Translate parts.
- Rotate parts.

- Scale parts.
- Mirror parts.
- Specify the coordinate system to use for transforming parts relative to the X, Y, or Z axes.
- Reset parts to their previous position, orientation, and size.

Note:

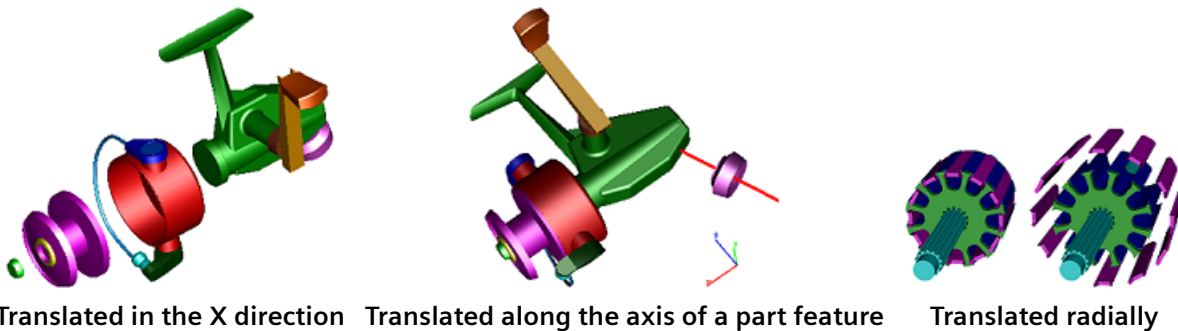
In the embedded viewer, the Teamcenter **Persistent Transformation** dialog displays the position of the selected assembly. When an assembly is selected, the **Feature** page is disabled.

Translating parts using the Part Transformation dialog box

Overview of translating parts using the Part Transformation dialog box

You can select parts and use the **Part Transformation** dialog box to translate or move them in the following ways:

- Translate parts along the X, Y, and Z axes.
- Translate parts along the axis of a selected part feature.
- Translate parts radially.



Translate parts along the X, Y, or Z axis

1. Select the parts to move.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the dialog box, click the **Transformation** tab.

The **Current Position (Global)** section shows the current global position of the selected part. (If you select more than one part, it shows the position of the first part.)

4. (Optional) Specify the coordinate system to use.
5. In the **Transform** section, click **Translate**.
6. Click **X**, **Y**, or **Z**.
7. (Optional) Specify the **Step** value.

The step value is the increment used by the **Delta** up and down arrows . The step value also controls the span of the slider. The slider span is 100 times the step value.

8. Do any of the following to translate the parts:
 - In the **Transform** section, move the slider to change the **Delta** value.
 - In the **Transform** section, type a **Delta** value and click **Apply**.
 - In the **Transform** section, use the arrows  to change the **Delta** value.
 - In the **Current Position (Global)** section, change the **Tx**, **Ty**, and **Tz** (translation) values.

Tip:

To translate along an axis, use the **Transform** section. Use the **Current Position (Global)** section only if you know the exact position (all 9 values) for the where the part should be.

Note:

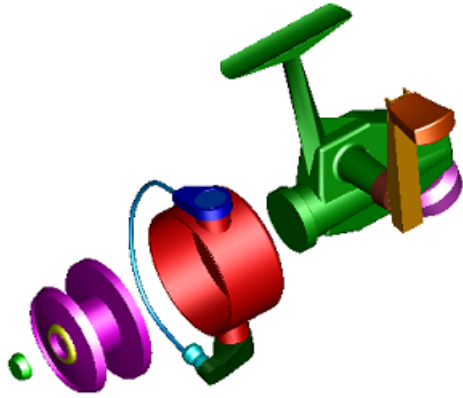
Changing the **Delta** value in the **Transform** section does a *relative* transformation and can change more than one value in the **Current Position (Global)** section.

9. (Optional) Do any of the following:
 - Continue transforming the parts.
 - To undo the last transformation that has not been applied, click **Undo**.
 - To undo all transformations that have not been applied, click **Reset All**.
 - To complete all transformations up to that point, click **Apply**.
 - Select other parts and transform them. This applies all active transformations.

- Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Example:

Several parts are translated in the X direction.



Translate parts along the axis of a part feature

- Select the parts to translate.
- From the **3D Part Transformation** toolbar, click **Transform parts** .
- In the **Part Transformation** dialog box, click the **Transformation** tab.
- Click **Pick Feature**, and then pick a **part feature** (edge or surface).

A line indicating the translation axis appears. The direction of the axis is determined by the part feature you pick. You can continue picking until you have the right feature.

- When you have the right feature, click **Stop Pick**.
- (Optional) In the **Translate** section, change the **Slider Span** value.

The value is the total number of units represented by the translation slider.

- Translate the selected parts along the axis in one of these ways:
 - Move the **Translate** slider.
 - Enter a **Translate** value and click **Apply**.

The selected part moves the specified number of units along the axis.

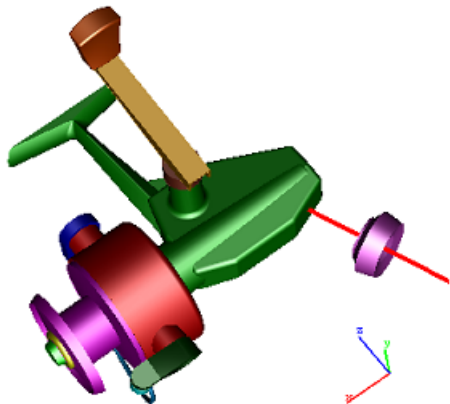
Note:

You can use the **Undo** command to undo the last transformation action.

8. (Optional) Do any of the following:
 - Continue transforming the parts.
 - To complete all transformations up to that point, click **Apply**.
 - Select other parts and transform them. This applies all active transformations.
9. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Example:

A part is translated along the axis of a part feature (the conical surface of the part).

**Translate parts radially**

1. Select the parts to move.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the **Part Transformation** dialog box, click the **Transformation** tab.
4. (Optional) Specify the coordinate system to use.
5. In the **Transform** section, click **Translate**.
6. Click **X**, **Y**, or **Z**.

7. Select the **Radial** check box.

8. (Optional) Specify the **Step** value.

The step value is the increment used by the **Delta** up and down arrows . The step value also controls the span of the slider. The slider span is 100 times the step value.

9. Move the selected parts radially in one of these ways:

- Move the slider to change the **Delta** value.
- Type a **Delta** value and click **Apply**.
- Use the arrows  to change the **Delta** value.

The selected parts move radially relative to the specified axis.

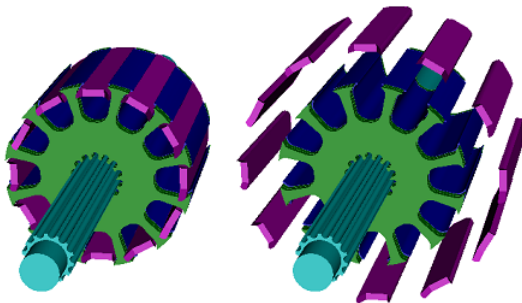
10. (Optional) Do any of the following:

- Continue transforming the parts.
- To undo the last transformation that has not been applied, click **Undo**.
- To undo all transformations that have not been applied, click **Reset All**.
- To complete all transformations up to that point, click **Apply**.
- Select other parts and transform them. This applies all active transformations.

11. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Example:

Several parts are translated radially.

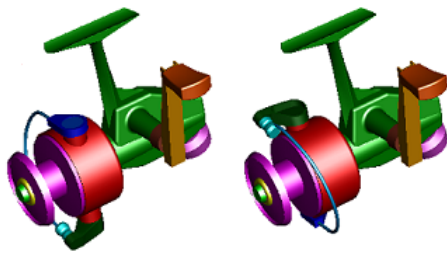


Rotating parts using the Part Transformation dialog box

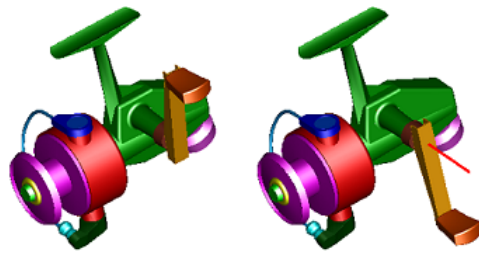
Overview of rotating parts using the Part Transformation dialog box

You can select parts and use the **Part Transformation** dialog box to rotate them.

- Rotate parts around the X, Y, or Z axis.
- Rotate parts around the axis of a part feature.



Spool assembly rotated around the X axis



Handle assembly rotated around the axis of a part feature

Rotate parts around the X, Y, or Z axis

1. Select the parts to rotate.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the dialog box, click the **Transformation** tab.

The **Current Position (Global)** section shows the current global position of the selected part. (If you select more than one part, it shows the position of the first part.)

4. (Optional) Specify the coordinate system to use.
5. In the **Transform** section, click **Rotate**.
6. Click **X**, **Y**, or **Z**.
7. (Optional) Specify the **Step** value.

The step value is the increment used by the **Delta** up and down arrows . The step value also controls the span of the slider. The slider span is 100 times the step value.

8. Do any of the following to rotate the parts:
 - In the **Transform** section, move the slider to change the **Delta** value.

- In the **Transform** section, type a **Delta** value and click **Apply**.
- In the **Transform** section, use the arrows  to change the **Delta** value.
- In the **Current Position (Global)** section, change the **Rx**, **Ry**, or **Rz** (rotation) value.

Tip:

To rotate around an axis, use the **Transform** section. Use the **Current Position (Global)** section only if you know the exact position (all 9 values) for the where the part should be.

Note:

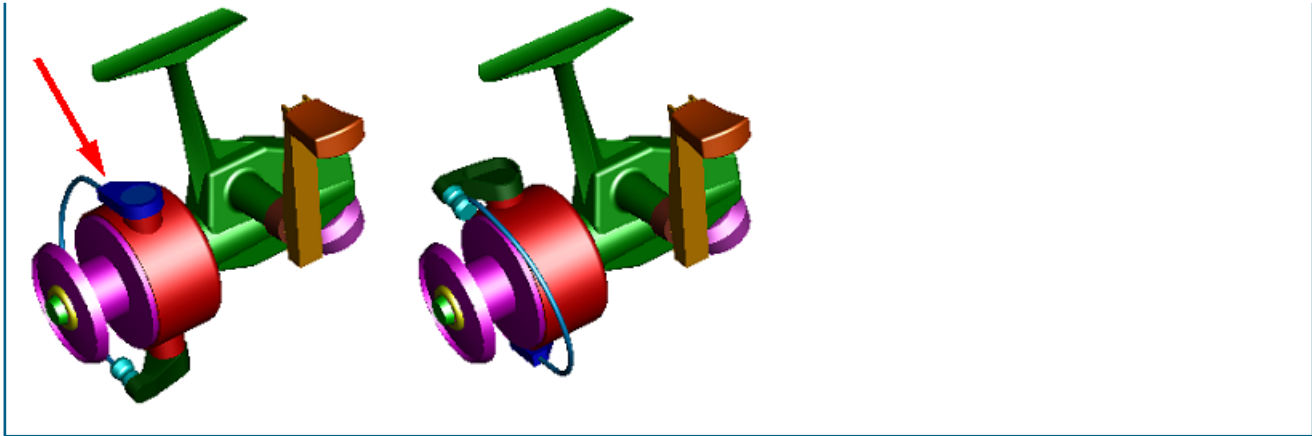
Changing the **Delta** value in the **Transform** section does a *relative* transformation and can change more than one value in the **Current Position (Global)** section.

9. (Optional) Do any of the following:

- Continue transforming the parts.
- To undo the last transformation that has not been applied, click **Undo**.
- To undo all transformations that have not been applied, click **Reset All**.
- To complete all transformations up to that point, click **Apply**.
- Select other parts and transform them. This applies all active transformations.

10. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.**Example:**

The spool assembly is rotated 180 degrees around the X axis which is the main axis of the model.



Rotate parts around the axis of a part feature

1. Select the parts to rotate.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the **Part Transformation** dialog box, click the **Feature** tab.
4. Click **Pick Feature**, and then pick a **part feature** (edge or surface).

A line indicating the rotation axis appears. The direction of the axis is determined by the part feature you pick. You can continue picking until you have the right feature.

5. When you have the right feature, click **Stop Pick**.
6. (Optional) In the **Translate** section, change the **Slider Span** value.

The value is the total number of degrees represented by the rotation slider.

7. Rotate the selected parts around the axis in one of these ways:
 - Move the **Rotate** slider.
 - Enter a **Rotate** value in the box and click **Apply**.

The selected part rotates the specified degrees around the axis.

Note:

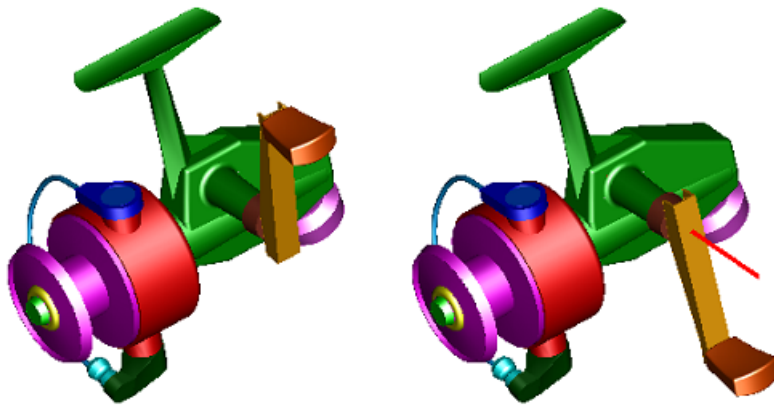
You can use the **Undo** command to undo the last transformation action.

8. (Optional) Do any of the following:

- Continue transforming the parts.
 - To complete all transformations up to that point, click **Apply**.
 - Select other parts and transform them. This applies all active transformations.
9. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Example:

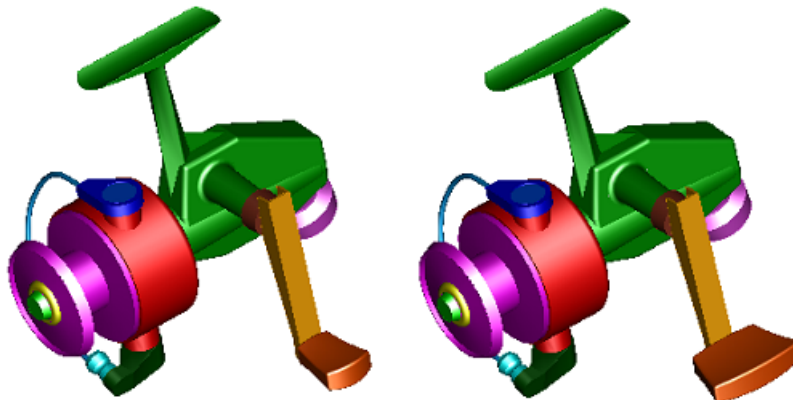
The handle subassembly is rotated 180 degrees around the axis of a selected part feature (conical surface of the boss).



Scaling parts using the Part Transformation dialog box

Overview of scaling parts

You can select parts and use the **Part Transformation** dialog box to scale them uniformly in all directions or relative to a single axis.



Handle part scaled in the Z direction

Scale parts

- 1. Select the parts to scale.
- 2. From the **3D Part Transformation** toolbar, click **Transform parts** .
- 3. In the dialog box, click the **Transformation** tab.

The **Current Position (Global)** section shows the current global position of the selected part. (If you select more than one part, it shows the position of the first part.)

- 4. (Optional) Specify the coordinate system to use.
- 5. In the **Transform** section, click **Scale**.
- 6. Do one of the following:

To	Do this
Scale the selected parts uniformly	Click the Uniform Scaling check box.
Scale the selected parts in X, Y, or Z	Clear the Uniform Scaling check box and click X , Y , or Z .

- 7. (Optional) Specify the **Step** value.

The step value is the increment used by the **Delta** up and down arrows . The step value also controls the span of the slider. The slider span is 100 times the step value.

- 8. Do any of the following to scale the parts:
 - In the **Transform** section, move the slider to change the **Delta** value.
 - In the **Transform** section, type a **Delta** value and click **Apply**.
 - In the **Transform** section, use the arrows  to change the **Delta** value.
 - In the **Current Position (Global)** section, change the **Sx**, **Sy**, or **Sz** (scale) value.

Note:
This option automatically clears the **Uniform Scaling** check box and scales the parts relative to the single axis.

Tip:

To scale about an axis, use the **Transform** section. Use the **Current Position (Global)** section only if you know the exact position (all 9 values) for the where the part should be.

Note:

Changing the **Delta** value in the **Transform** section does a *relative* transformation and can change more than one value in the **Current Position (Global)** section.

9. (Optional) Do any of the following:

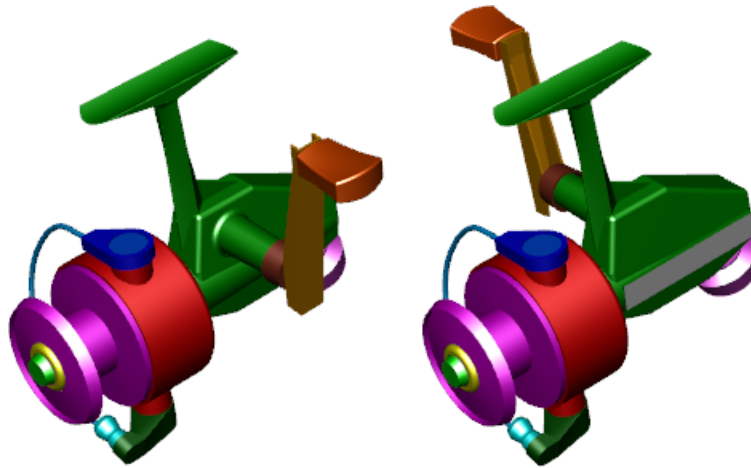
- Continue transforming the parts.
- To undo the last transformation that has not been applied, click **Undo**.
- To undo all transformations that have not been applied, click **Reset All**.
- To complete all transformations up to that point, click **Apply**.
- Select other parts and transform them. This applies all active transformations.

10. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Mirroring parts using the Part Transformation dialog box

Overview of mirroring parts

You can select parts and use the **Part Transformation** dialog box to mirror them relative to the XY, XZ, or YZ plane.



Body and handle mirrored

Mirror parts

1. Select the parts to mirror.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the dialog box, click the **Transformation** tab.

The **Current Position (Global)** section shows the current global position of the selected part. (If you select more than one part, it shows the position of the first part.)

4. (Optional) Specify the coordinate system to use.
5. If **Translate** is active, clear the **Radial** check box.
If **Scale** is active, clear the **Uniform Scaling** check box.
6. Click **X**, **Y**, or **Z** to specify the normal of the mirror plane.
7. Click **Mirror**.

8. (Optional) Do any of the following:
 - Continue transforming the parts.
 - To undo the last transformation that has not been applied, click **Undo**.
 - To undo all transformations that have not been applied, click **Reset All**.
 - To complete all transformations up to that point, click **Apply**.

- Select other parts and transform them. This applies all active transformations.
9. Click **OK** to apply all active transformations and close the **Part Transformation** dialog box, or click **Cancel** to cancel active transformations and close the dialog box.

Specify a coordinate system for transformations

Specify the coordinate system to use for transforming parts relative to the X, Y, or Z axes.

1. Select the parts to transform.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the **Part Transformation** dialog box, click the **Transformation** tab.
4. In the **Coordinate System** section, select one of the following from the list:

Select	To use
Global	The Global coordinate system.
Screen	The X, Y, and Z axes of the computer screen with the center of rotation in the middle of the screen.
Selected Part	The coordinate system that was defined when the selected part was created. If you select multiple parts, it is the coordinate system of the first part you select.
Selected Part's Bounding Box	A coordinate system with its origin in the center of the bounding box of the selected part. The axis orientation is the same as the part coordinate system. If you select multiple parts, it uses the bounding box of the first part you select.
Working Coordinate System	A User Defined or PMI coordinate system that is designated as the Working CS. If you have not already designated a Working CS, you can choose a coordinate system in the Select Coordinate System dialog box.

Note:

You can select multiple parts transform. Selected Part and Selected Part's Bounding Box coordinate systems use the first part you select.

5. (Optional) If you select **Working Coordinate System**, you can do the following:
 - Click **Change** and select another coordinate system to be the Working Coordinate System.

- Click **Manage** to open the **Manage Coordinate System** dialog box where you can create and edit coordinate systems, and choose a coordinate system to be the Working CS.
6. (Optional) To enable 6-DOF input devices to move parts using the coordinate system you selected above, instead of the Screen Coordinate System, clear the **Use Screen Coordinates for 6-DOF input devices** check box.
 7. (Optional) Select the **Set as default** check box to make your selected coordinate system the default that appears each time you use Part Transformation. You cannot set the Working Coordinate System to be the default.

Reset parts to their original position, orientation, and size

1. Do the following:
 - If you transformed individual parts, select the parts to reset.
 - If you transformed assemblies, select the assemblies to reset.
2. From the **3D Part Transformation** toolbar, click **Transform parts** .
3. In the **Part Transformation** dialog box, click the **Transformation** tab.
4. Click **Reset All**.

Using manipulators to transform parts

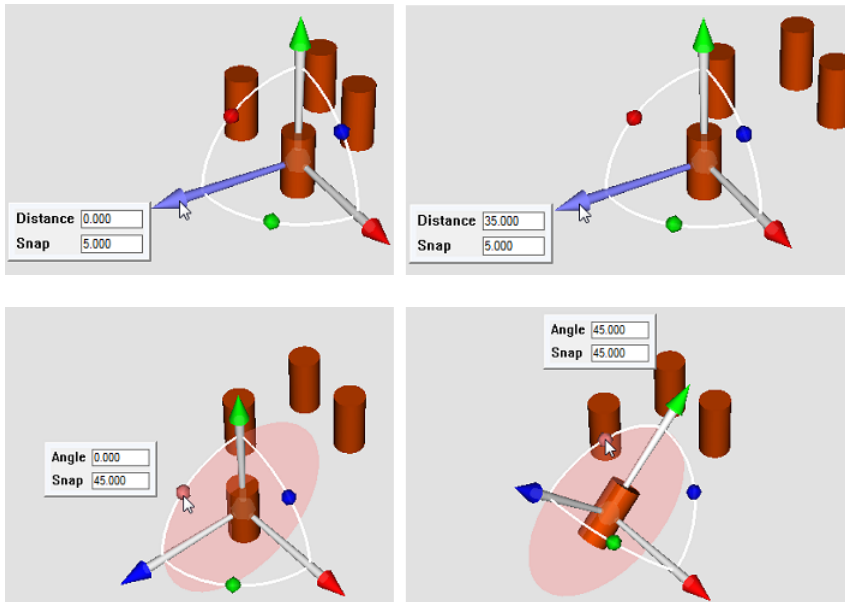
Overview of using manipulators to transform parts

Interactive part manipulators are one of several tools you can use to **transform parts**. You can add a manipulator to parts and then use the manipulator to translate and rotate the parts. When you move the manipulator, the parts move with it. Part manipulators appear as interactive triads. They work the same as coordinate system manipulators.

You can do the following with manipulators:

- Drag to translate and rotate.
- Enter delta angle or distance values.
- Enter X, Y, Z coordinates.
- Align to coordinate systems.
- Snap to feature geometry.

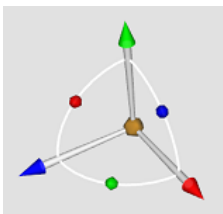
- Align a manipulator axis to an edge, or a manipulator plane to a surface.



Use manipulators to transform parts

1. Select the parts to transform.
2. On the **3D Part Transformation** toolbar, click **Add Manipulator** .

A manipulator appears around the part. The manipulator is an interactive triad with three axis arrows for translating and three small spheres for rotating.



3. (Optional) Repeat the previous steps to add manipulators to other parts.
4. If you have more than one manipulator, click a manipulator to activate it.

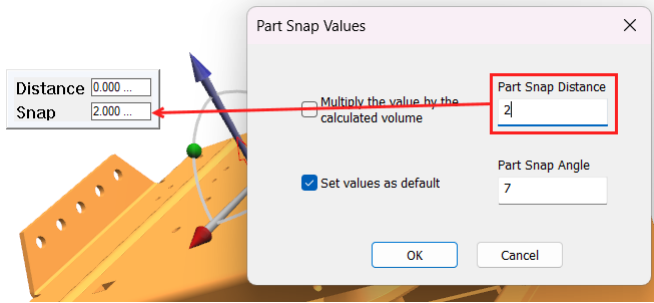
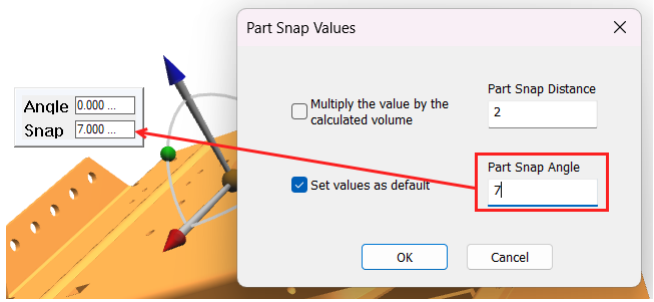
The active manipulator has brighter colors. Only one manipulator is active at any time.

5. Transform the parts in any of these ways:

To	Do this						
Drag the parts in X, Y, or Z	Drag the X, Y, or Z manipulator axis. (Red is X, green is Y, and blue is Z.)						
Translate parts precise delta distances in X, Y, or Z	a. If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click the X, Y, or Z manipulator axis to display the distance window. c. Do one of the following: <table border="1"> <thead> <tr> <th>To</th><th>Do this</th></tr> </thead> <tbody> <tr> <td>Drag using a snap increment</td><td> A. (Optional) Type a new Snap value in the distance window. B. Drag the axis. If Snap is set to 5, the parts move in 5 unit increments. </td></tr> <tr> <td>Specify a delta distance</td><td> Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value. </td></tr> </tbody> </table>	To	Do this	Drag using a snap increment	A. (Optional) Type a new Snap value in the distance window. B. Drag the axis. If Snap is set to 5, the parts move in 5 unit increments.	Specify a delta distance	Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.
To	Do this						
Drag using a snap increment	A. (Optional) Type a new Snap value in the distance window. B. Drag the axis. If Snap is set to 5, the parts move in 5 unit increments.						
Specify a delta distance	Enter the Distance in the distance window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.						
Translate parts by typing their new X, Y, Z coordinates	a. If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click one of the manipulator arcs or the sphere at the manipulator origin to display the position window. c. Enter new X, Y, Z coordinate values.						
Rotate parts around an axis	Drag one of the small spheres.						
Rotate parts a specific number of degrees around an axis	a. If the manipulator window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click one of the small spheres. c. Do one of the following:						

To	Do this	

To	Do this							
	<table><tr><th>To</th><th>Do this</th></tr><tr><td>Align a manipulator axis to an edge on the model</td><td>a. Right-click the manipulator axis and choose Align Axis. b. Click the edge.</td></tr><tr><td>Align a manipulator plane to a surface on the model</td><td>a. Right-click the small sphere on the manipulator plane and choose Align Plane. b. Click the surface.</td></tr></table>	To	Do this	Align a manipulator axis to an edge on the model	a. Right-click the manipulator axis and choose Align Axis . b. Click the edge.	Align a manipulator plane to a surface on the model	a. Right-click the small sphere on the manipulator plane and choose Align Plane . b. Click the surface.	
	To	Do this						
	Align a manipulator axis to an edge on the model	a. Right-click the manipulator axis and choose Align Axis . b. Click the edge.						
	Align a manipulator plane to a surface on the model	a. Right-click the small sphere on the manipulator plane and choose Align Plane . b. Click the surface.						
The parts rotate to align to the edge or surface.								
Rotate around a different center of rotation, or translate along reorientated axes	a. Right-click the manipulator and turn off Attached. The manipulator is no longer attached to the parts. b. Translate and rotate the manipulator. The parts do not move. c. Right-click the manipulator and turn on Attached to reattach the manipulator to the parts. d. Continue translating and rotating the parts with the manipulator.							
Undo the last transformation action	Press Ctrl+Z. Note: To reverse an undo, press Ctrl+Y.							
Set the default snap value for distance and angles	a. Right-click the manipulator and choose Snap Values... The Part Snap Values dialog box displays. b. Do any of the following. <table><tr><th>To</th><th>Do this</th></tr><tr><td>Define a specific default</td><td>Enter a default snap distance value in the Part Snap Distance field.</td></tr></table>		To	Do this	Define a specific default	Enter a default snap distance value in the Part Snap Distance field.		
To	Do this							
Define a specific default	Enter a default snap distance value in the Part Snap Distance field.							

To	Do this
distance snap value.	
Multiply the system-calculated part volume by a specific value for the distance snap value.	Enter a default snap distance value in the Part Snap Distance field and select the Multiply the value by the calculated volume checkbox.
Use only the system-calculated part volume as the distance snap value.	Enter 1 in the Part Snap Distance field and select the Multiply the value by the calculated volume checkbox.
Define a specific default angle distance snap value.	

To	Do this
	c. (Optional) To save these values in the registry for use with other models, select Set values as Default. d. Click OK.
Set the part manipulator to the part origin	Right-click the manipulator and choose Set Manipulator to Part Origin. Choose Set Manipulator to Part Origin again to move the manipulator back to the center of the part's bounding box.

6. (Optional) To return the active manipulator and parts to their original position, on the **3D Manipulators** toolbar, click **Reset Manipulator Position** .

Tip:

You can also use **Reset All** in the **Part Transformation** dialog box to reset all selected parts.

7. To remove a manipulator, click **Remove Manipulator**  on the **3D Part Transformation** toolbar.

Note:

If you add a manipulator to a set of parts, transform the parts, and then choose **Reset Manipulator Position**, the parts and manipulator return to their original positions. Resetting can become complicated when you add parts to the original set of parts and add another manipulator. For example, if you:

1. Select one or more parts, add a manipulator, and transform the parts. (The parts are considered to be the master parts.)
2. With the original (master) parts still selected, select additional parts and add a second manipulator.
3. Use the second manipulator to transform all the selected parts.
4. Choose **Reset Manipulator Position**.

The master parts and manipulator return to their original positions, but the parts you selected later do not. They keep their same relative positions to the master parts.

If necessary, you can use **Reset All** in the **Part Transformation** dialog box to reset all selected parts.

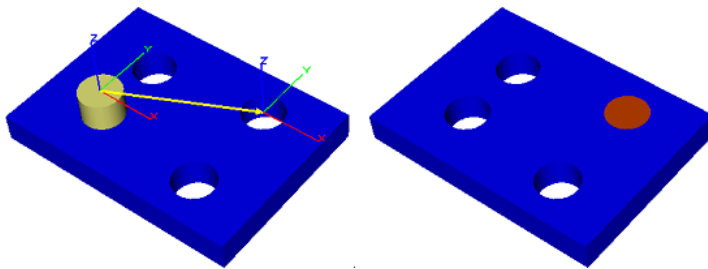
Repositioning parts relative to other parts

Overview of repositioning parts relative to other parts

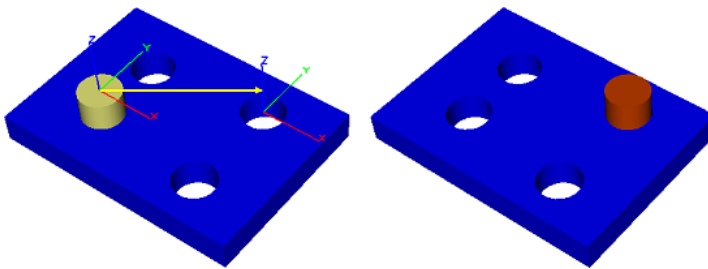
The **Relocate** tool is one of several tools you can use to **transform parts**. You can use **Relocate** to transform parts from one location to another by specifying a *From Frame* and a *To Frame*. You can also restrict movements to one or two axes, and translate parts without changing their orientation.

After selecting the parts, you specify the From Frame and To Frame. The From Frame is the handle used to grab the parts. (It does not have to be on the parts.) The To Frame is where you let go of the handle. Temporary coordinate systems appear at the from and to points. You can manipulate both coordinate systems before applying the transformation.

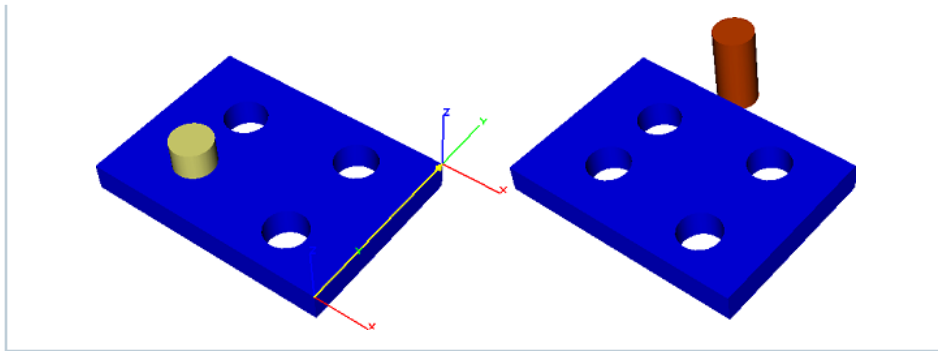
In this example, the From Frame is the top of the highlighted part. The To Frame is the center of the hole. Temporary coordinate systems appear at each point. The first coordinate system (and the part) is transformed to the second coordinate system.



This example is the same as the first example, except the translation is limited to two axes.



In this example, the From Frame is a corner of the block. The To Frame is another corner of the block. Neither point is on the part to be transformed. The first coordinate system (and the part) is transformed to the second coordinate system.



Use Relocate to reposition parts relative to other parts

1. Select the parts to relocate.
2. On the **3D Part Transformation** toolbar, click **Relocate parts** .
3. In the **Relocate** dialog box, from the **From Frame** list, choose one of the following to specify the From Frame:

To	Do this
Use the coordinate system of the parts	Choose Self .
Use the center of the bounding box of the selected parts	Choose Geometric Center .
Select a part feature	a. Choose Pick Feature . b. Select the feature .

A temporary coordinate system appears at the From Frame location.

4. (Optional) Select the **Manipulator** check box for the From Frame, and use the manipulator to reposition the From Frame coordinate system. Clear the check box to remove the manipulator.
5. From the **To Frame** list, choose one of the following to specify the To Frame:

To	Do this
Use the coordinate system of the parts	Choose Self .
Use the center of the bounding box of the selected parts	Choose Geometric Center .
Select a part feature	a. Choose Pick Feature . b. Select the feature .
Select a point in space	a. Choose Pick Feature .

To	Do this
	b. Pick a point not on a part. c. Click Stop Picking .

A temporary coordinate system appears at the To Frame location, and a yellow arrow appears to show how the parts will move.

6. (Optional) Select the **Manipulator** check box for the From Frame or To Frame and then use the manipulator to reposition the coordinate system. Clear the check box to remove the manipulator.
7. (Optional) Do any of the following to constrain the transformation:

To	Do this
Translate the parts without changing their current orientation	Select the Maintain Orientation check box.
Limit the translation to only one or two axes	a. Select the Translate only on check box. b. Select one or two of the X , Y , or Z check boxes.

8. Click one of the following:

Use this option	To do this
Apply	The transform the parts without closing the dialog box.
Reset	Return the parts to their original positions without closing the dialog box.
Flip	Rotate the parts 180 degrees around the X axis of the To Frame by reversing the Z axis, after applying the transformation.
OK	Transform the parts and close the dialog box.
Cancel	Close the dialog box before applying current the transformation.

Tip:

You can use the **Reset All** option in the **Part Transformation** dialog box to reset parts to their original positions after closing the **Relocate** dialog box.

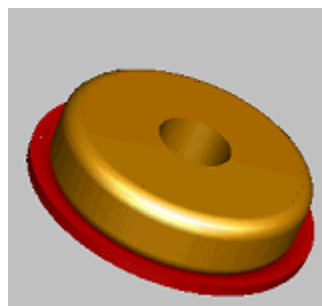
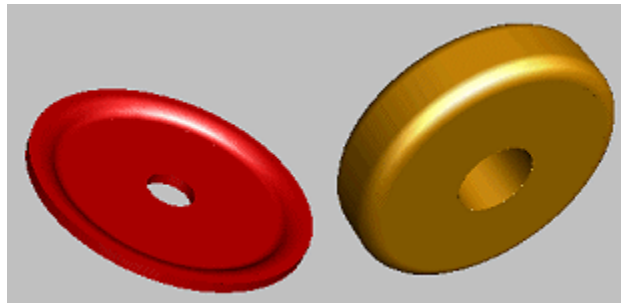
Aligning parts

Overview of aligning parts

Use part Alignment tools to move and rotate selected parts so they align with other parts in the Viewing window.

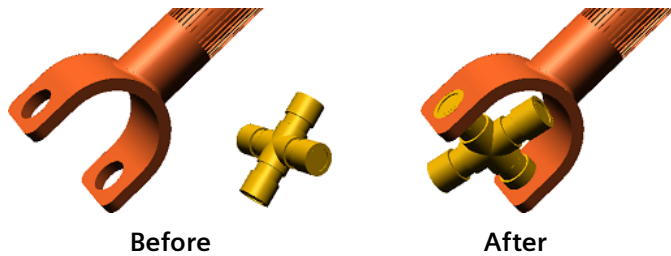
You can

- Move individual parts or groups of parts and align them to other parts in the Viewing window.
- Reset parts to their previous positions after accepting the alignment.
- Set part alignment preferences.



Align parts with other parts

Align parts with other parts by defining up to three alignment constraints. Preview possible alignments before accepting the correct one.



1. On the **3D Alignment** toolbar, click **3D Alignment** .
2. **Select** the part or parts to move.
3. On the **3D Alignment** toolbar, click **Primary Constraints** .
4. On the **3D Selection** toolbar, specify the part feature **type** (point, vertex, edge, or surface) of the first feature.
5. In the Viewing window, pick the first **part feature**.

The part feature changes color, and a trihedron appears. The letter **A** in a circle identifies it as the first feature in the primary constraint.

Note:

The first pick can be a feature on any part. When you preview and accept the alignment, only the selected parts will move. Picking a feature on a part that is not selected may result in an unexpected alignment.

6. (Optional) Specify a different feature type of the second feature.
7. In the Viewing window, pick the second part feature.

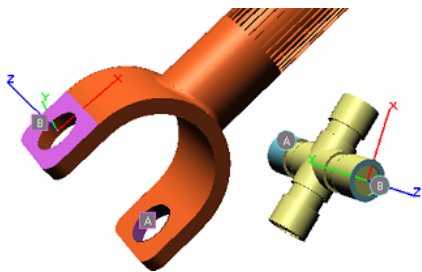
The feature changes color, and a trihedron appears. The letter **A** in a square identifies it as the second feature in the primary constraint.

8. (Optional) To define a second constraint, click **Secondary Constraints**  and pick two part features.

The features changes color, and trihedrons appear. The letter **B** in the circle and square identifies them as parts of the secondary constraint.

9. (Optional) To define a third constraint to further constrain the alignment, click **Tertiary Constraints**  and pick two part features.

The features changes color, and trihedrons appear. The letter **C** in the circle and square identifies them as parts of the tertiary constraint.



Primary (A) and secondary (B) constraints

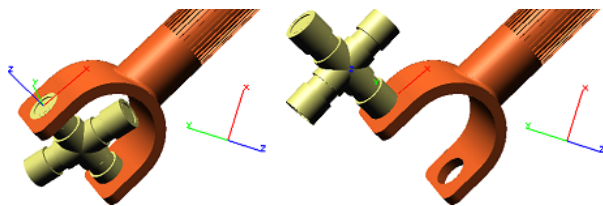
10. Preview the alignment.

- a. On the **3D Alignment** toolbar, click **Preview Alignment** .

The selected parts move to align with the target parts.

- b. Click **Next Alignment**  to cycle through the possible alignments.

There may be multiple ways for the part to satisfy the constraints.



Alignment preview 1

Alignment preview 2

11. Do one of the following:

To	Do this
Accept the alignment.	On the 3D Alignment toolbar, click Accept Alignment  .
	Note: Parts are aligned according to the part specifications. If your model is displayed at a low level of detail, parts may not align as you expect.
Stop the preview and refine constraints.	a. On the 3D Alignment toolbar, click Preview Alignment  .

To	Do this
	The selected parts return to their original positions. b. Repeat the steps above to define the constraints and preview the alignment.
Cancel the alignment.	On the 3D Alignment toolbar, click Cancel Alignment . The selected parts return to their original positions.

Reset parts to their original positions after accepting alignment

1. Align two or more parts.
2. Do one of the following:

To	Do this
Undo the last alignment action	Press Ctrl+Z. Note: To reverse an undo, press Ctrl+Y.
Reset selected parts to their original positions	Use the Reset All option in the Part Transformation dialog box.

Set alignment preferences

1. Choose **Alignment**→**Preferences**.
2. In the **Alignment Preferences** dialog box, do any of the following:

To	Do this
Change the color of picked part features, labels, and leader lines in the part you are moving.	From the Source Coordinate System color list, select a color.
Change the color of picked part features, labels, and leader lines in the part to which you are aligning.	From the Destination Coordinate System color list, select a color.

To	Do this
Change the size of the trihedrons.	Move the Trihedron Size slider to the left to decrease the size, or to the right to increase the size.
Change the width (pixels) of the lines in the trihedrons.	From the Trihedron Line Width list, select a width value.
Display or hide the trihedrons at the source and destination coordinates.	Select or clear the Draw Trihedrons check box.

Transforming parts by dragging

Overview of transforming parts by dragging

Part Manipulation Mode is one of several tools you can use to **transform parts**. Each tool works in a slightly different way. With **3D Part Manipulation Mode** active, you translate (move) parts by dragging them around the Viewing window. You translate using the Screen, Global, or Part coordinate systems. You can also reset the parts to their original positions.

Depending upon the chosen coordinate system, you can translate your parts in the following ways:

Use this option	To do this
X	Translate selected parts along the X axis.
Y	Translate selected parts along the Y axis.
Z	Translate selected parts along the Z axis. (Not available when using the Screen coordinate system.)
XY	Translate selected parts on the XY plane.
XZ	Translate selected parts on the XZ plane. (Not available when using the Screen coordinate system.)
YZ	Translate selected parts on the YZ plane. (Not available when using the Screen coordinate system.)

Drag parts around the Viewing window

1. On the **3D Part Manipulation** toolbar, click **Enable Part Manipulation Mode** .
2. Do any of the following:

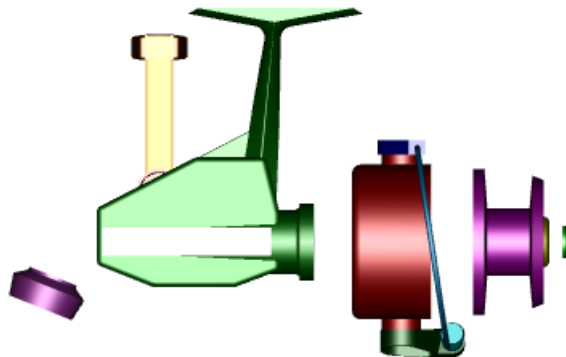
To	Do this
Move parts interactively using the screen coordinate system	a. On the 3D Part Manipulation toolbar, click Use Screen Coordinate System . b. On the 3D Part Manipulation toolbar, click X , Y , or XY .
Move parts interactively using the global coordinate system	a. On the 3D Part Manipulation toolbar, click Use Global Coordinate System . b. On the 3D Part Manipulation toolbar, click X , Y , Z , XY , XZ , or YZ .
Move parts interactively using the part coordinate system	a. On the 3D Part Manipulation toolbar, click Use Part Coordinate System . b. On the 3D Part Manipulation toolbar, click X , Y , Z , XY , XZ , or YZ . Note: The part selected first designates the part coordinate system used.
Undo translations	Press Ctrl+Z. Note: To reverse an undo, press Ctrl+Y.
Return parts to their original positions	a. Select the parts to be reset to their original positions. b. On the 3D Part Manipulation toolbar, click Reset to Original Position . All transformed parts are returned to their original position. However, if you moved an assembly and the parts to be reset are the sub-parts of the assembly, the parts do not return to their original position since they were not moved relative to the assembly. To reset these parts, select the assembly and

To	Do this
	perform Step b above. The assembly returns to its original position. Tip: To reset <i>all</i> parts in your model to original position, choose Actions→Motion→Reset All Parts.

3. In the Viewing window, select one or more parts and drag them to new positions.
4. To exit **Part Manipulation Mode**, on the **3D Part Manipulation** toolbar, click **Enable Part Manipulation Mode**  again.

Create exploded views

Create exploded views, in which parts and groups of parts are separated by a specified distance along an axis.



1. Choose **Actions→Exploded View**.
2. In the **Exploded View** dialog box, click **New**.
3. In the **Name** box, type a name for the exploded view, and click **OK**.

Note:

The name must be unique to this work session, but need not be unique among all work sessions.

4. To add parts to the exploded view, do the following:

To	Do this
Add a part	Select a part, and click Add .
Add multiple parts	Select the parts, and click Add As Group .
Move a part or group of parts up or down within the Ordered part list	a. In the Ordered part list, select the part or group of parts. b. Click Move Up or Move Down. Note: The first parts in the Ordered part list function as the base of the exploded view. All other parts in the list move with respect to the base parts.
Create a group of parts from individual parts in the Ordered part list	a. In the Ordered part list, select the parts that you want to group. b. Click Group.
Separate a group of parts into individual parts in the Ordered part list	a. In the Ordered part list, select the parts that you want to separate. b. Click Ungroup.
Remove a part or group from the Ordered part list	a. In the Ordered part list, select the parts that you want to remove. b. Click Remove.

5. In the **Axis** section, select the axis along which you want the parts to explode.

-or-

In the **Axis** section, click **Pick**, and then use the **3D Selection** toolbar to **pick** a part surface or edge.

6. In the **Direction** section, select a direction for the parts to explode along the specified axis.
7. (Optional) To display the centerline along which the parts explode, select **Draw Centerline**.

Note:

To change the style, color, or thickness of the center line, right-click the line, and click **Properties**.

8. In the **Distance Between Parts** box, type the distance, in model units, to separate the parts.

Note:

When the parts are exploded, parts move the specified distance from the bounding box of the previous part on the list.

9. Click **Apply**.

The parts in the list move the specified number of units along the axis to create an exploded view.

10. (Optional) To return parts to their original positions, click **Reset All**.
11. (Optional) To delete an exploded view, from the **Available explosions** list, select the exploded view, and click **Delete**.

6. Constraining assemblies

Overview of constraints

Constraints control the position of assembly components by defining relationships between component geometry.

Using constraints, you can simulate mechanisms involving components from the same or different subassemblies.

Associative constraints

Associative constraints remember the relationships between component geometry. These relationships are honored when you play motion files or manually transform parts. For example, if you create a Touch constraint between faces on two parts and then move one part, the second part moves with it.

In Mockup, to create associative constraints, your JT models must contain B-Rep Entity IDs. Entity IDs are unique numbers assigned to edges and faces when a CAD model is translated. You can use **Inspector to determine if a JT file contains B-Rep Entity IDs**, and therefore support authoring associated constraints. Not all models with B-Rep data have B-Rep Entity IDs.

In Mockup, use associative constraints to:

- Constrain components so they are (and must remain) tangent, coincident, parallel, or perpendicular.
- Specify that a component is fixed in place.
- Specify a distance or angle that must be maintained between components.
- Constrain components so they move as a single rigid unit.

In Mockup, if you create associative constraints, you can:

- Save and load constraint files.
- Manage constraints in the Constraints view.
- View and manipulate constraint labels in the Viewing window.
- Show the degrees of freedom for constrained components.

Positional constraints

In Mockup, if your models contain B-Rep data, but not Entity IDs, you can create positional constraints to position components in an assembly. You cannot save positional constraints.

You can use positional constraints to:

- Position components so they are tangent, coincident, parallel, or perpendicular.
- Position components so there is a specified distance or angle between them.
- Specify that a component is fixed in place so it does not move when you reposition other components.
- Position components so they move as a single rigid unit.

Constraint types

Constraint type	Description
 Touch	Constrains two components such that the selected two geometric features are tangent or coincident depending on the geometry.
 Distance	Specifies the distance between two components.
 Fix	Fixes a component at its current position such that it has zero degrees of freedom.
 Parallel	Constrains two components such that the selected two geometric features are parallel.
 Perpendicular	Constrains two components such that the selected two geometric features are perpendicular.
 Angle	Constrains two components such that the selected two geometric features are at a specified angle.
 Bond	Constrains two or more components so they move as a single rigid unit.

Creating constraints

Workflow for creating associative constraints

The following describes the workflow for creating any type of associative constraint.

Constraints are created between subassemblies of selected part/leaf components. This provides the ability to move entire subassemblies/leaf components as a rigid set of geometries. By default, constraints

are created between subassemblies, not between parts. To move individual components in a rigid subassembly, you can define them as flexible moveable components in the system. You change rigid subassembly components to flexible moveable components to simulate a mechanism, using the **mechanism feature**.

1. Enable **Associative** constraints, and then specify the constraint type.
2. Select the components to fix or the component geometry to constrain.
3. Depending on the type of geometry selected, refine the selection.
4. For Distance and Angle constraints, specify the distance or angle value.
5. If the **Honor Constraints** preference is set, the constraint is solved and the components move into their constrained positions. If necessary, click **Reverse** or **Cycle** to see other possible positioning.
6. Apply the constraint.

Create Touch constraints

Touch constraints constrain two components so the selected geometry features are tangent or coincident. You select two components and specify a geometric feature for each. If one of the features is curved, you create a tangent constraint. If neither feature is curved, you create a coincident constraint.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	Do this
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, make sure **Associative** is selected.
5. (Optional) Use the **Components to Move** option if you are creating a constraint to move a group of components together as a bond constraint and mechanism.
6. From the **Constraint Type** list, select **Touch**.
7. Click **Select two objects (0)**.

8. Select the first part feature (edge or surface).

Note:

During part solving, the first selected component is moved and the second part is fixed. If the first component is fixed, then the second selected component moves.

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

9. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to **Create Constraints** dialog box, click **Cancel**.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.
- d. Click **OK**.

10. Select the second part feature (edge or surface).

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

11. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to the **Create Constraints** dialog box, click **Cancel**.

You must click **Select two objects (0)** again to select a second component.

If you choose a different constraint type, the **Create Constraints** dialog box is reset to its initial state.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.

- d. Click **OK**.

12. (Optional) If available, select **Reverse** to see other possible positioning. This option is available only for constraints between some part features.

13. Click **Apply**.

A Touch constraint label  appears near the components.

Create Distance constraints

Distance constraints constrain two components so the selected geometry features are a specified distance from each other. You select two components and specify a geometric feature for each.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, make sure **Associative** is selected.
5. (Optional) Use the **Components to Move** option if you are creating a constraint to move a group of components together as a bond constraint and mechanism.
6. From the **Constraint Type** list, select **Distance**.
7. Click **Select two objects (0)**.
8. Select the first part feature (edge or surface).

Note:

During part solving, the first selected component is moved and the second part is fixed. If the first component is fixed, then the second selected component moves. However, you can use the **Reposition** options to choose the component to move.

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

9. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to **Create Constraints** dialog box, click **Cancel**.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.

- d. Click **OK**.

10. Select the second part feature (edge or surface).

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

11. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to the **Create Constraints** dialog box, click **Cancel**.

You must click **Select two objects (0)** again to select a second component.

If you choose a different constraint type, the **Create Constraints** dialog box is reset to its initial state.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.
- d. Click **OK**.

12. Type the **Distance** value.

13. In the **Reposition** section, use the slider to adjust the distance between the components.

By default, the slider adjusts the value in increments of 1 measure unit. To increase the increment value, type it in the **Step** box.

14. (Optional) If available, select **Reverse** or **Cycle** to see other possible positioning. These options are available only for constraints between some part features.
15. Click **Apply**.

A Distance  constraint label appears near the components.

Create Fix constraints

Fix constraints lock components in place. Fixed components have zero degrees of freedom.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, make sure **Associative** is selected.
5. From the **Constraint Type** list, select **Fix**.
6. Click **Select objects (0)**.
7. In the Viewing window or the assembly tree, select the component to fix.

(Optional) To select multiple components, press the Ctrl key as you select the components.
8. Click **Apply**.

A Fix constraint label  appears next to the each fixed component.

Create Parallel or Perpendicular constraints

Parallel and Perpendicular constraints constrain two components so the selected geometry features are parallel or perpendicular. You select two components and specify a geometric feature for each.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, make sure **Associative** is selected.
5. (Optional) Use the **Components to Move** option if you are creating a constraint to move a group of components together as a bond constraint and mechanism.
6. From the **Constraint Type** list, select **Parallel** or **Perpendicular**.
7. Click **Select two objects (0)**.
8. Select the first part feature (edge or surface).

Note:

During part solving, the first selected component is moved and the second part is fixed. If the first component is fixed, then the second selected component moves.

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

9. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to **Create Constraints** dialog box, click **Cancel**.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

c. Click **Apply**.

d. Click **OK**.

10. Select the second feature (edge or surface).

11. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to the **Create Constraints** dialog box, click **Cancel**.

You must click **Select two objects (0)** again to select a second component.

If you choose a different constraint type, the **Create Constraints** dialog box is reset to its initial state.

a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

c. Click **Apply**.

d. Click **OK**.

12. (Optional) If available, select **Reverse** to see other possible positioning. This option is available only for constraints between some part features.
13. Click **Apply**.

A Parallel  or Perpendicular  constraint label appears near the components.

Create Angle constraints

Angle constraints constrain two components so the selected geometry features are at a specified angle to each other. You select two components and specify a geometric feature for each.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, make sure **Associative** is selected.
5. (Optional) Use the **Components to Move** option if you are creating a constraint to move a group of components together as a bond constraint and mechanism.
6. From the **Constraint Type** list, select **Angle**.
7. Click **Select two objects (0)**.
8. Select the first part feature (edge or surface).

Note:

During part solving, the first selected component is moved and the second part is fixed. If the first component is fixed, then the second selected component moves. However, you can use the **Reposition** options to choose the component to move.

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

9. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to **Create Constraints** dialog box, click **Cancel**.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. U and V values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.

- d. Click **OK**.

10. Select the second feature (edge or surface).

If you select a non-planar surface or a non-straight edge, the **Selection Filter** dialog box appears to help you narrow the selection to the face/axis/point on the face or the edge/center/point on the edge.

11. If the **Selection Filter** dialog box appears, do the following:

Note:

To cancel the part feature selection and return to the **Create Constraints** dialog box, click **Cancel**.

You must click **Select two objects (0)** again to select a second component.

If you choose a different constraint type, the **Create Constraints** dialog box is reset to its initial state.

- a. From the **Selection Type** list, select one of the following:

Select	To specify
Face	The curved surface for tangency.
Axis	The axis of the curved surface.
Point on face	A point on the curved surface.
Edge	The curved edge for tangency.
Center	The center of the curved edge.
Point on Edge/Curve	A point on the curve.

- b. (Optional) If you select **Point on Face** or **Point on Edge/Curve**, specify the point in the **Location on Face/Edge** or **Output Coordinates** sections.

The **U** parameter specifies the location along the primary direction of the curved surface or the edge. **V** specifies the location along the other direction of the surface. **U** and **V** values range from 0 to 1, with 0.5 being the middle. **X**, **Y**, and **Z** are the coordinates of the point.

- c. Click **Apply**.
- d. Click **OK**.

12. Do one of the following:

To	Do this
Specify an axis about which the angle is measured.	a. Click Planar Angle. b. Click Select Axis. c. In the Viewing window, select a part feature (line, plane, cylinder, or swept surface).

To	Do this
	d. Type the Angle value.
Constrain without specifying an axis.	a. Click 3D Angle . b. Type the Angle value.

13. In the **Reposition** section, use the slider to adjust the angle between the components.

By default, the slider adjusts the value in increments of 1 degree. To increase the increment value, type it in the **Step** box.

14. (Optional) For a 3D angle constraint, select **Reverse** to see other possible positioning.

15. Click **Apply**.

An Angle  constraint label appears near the components.

Create Bond constraints

Bond constraints constrain two or more components so they move as a single rigid unit. Components of a Bond constraint must have the same parent, and a component may belong to only one bond constraint.

- From the **3D Constraints** toolbar, click **Constraint** .
- If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

- From the **3D Constraints** toolbar, click **Create Constraint** .
- In the **Create Constraints** dialog box, make sure **Associative** is selected.
- From the **Constraint Type** list, select **Bond**.
- Click **Select objects (0)**.

7. Press the Ctrl key, and in the Viewing window or the assembly tree, select the components to bond.
8. Click **Apply**.

A Bond  constraint label appears near the components.

Creating constraints between subassemblies

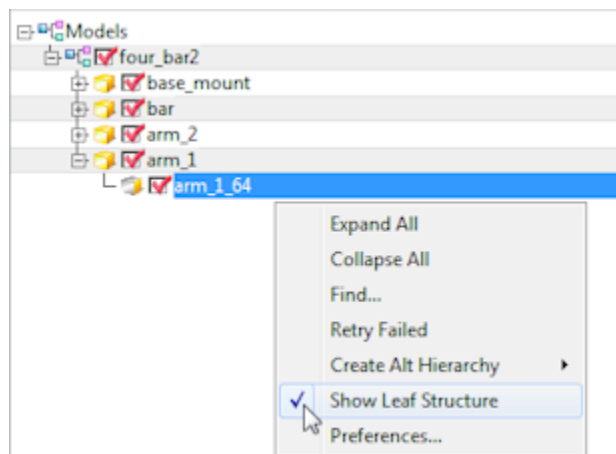
Overview of assembly positioning

When you create a constraint selecting parts or leaf nodes from different subassemblies without a mechanism, the constraint is created between the subassemblies and placed at the nearest common node in the assembly tree. The entire subassembly is constrained as a rigid set.

To move individual components in a subassembly, you must define them as a mechanism component.

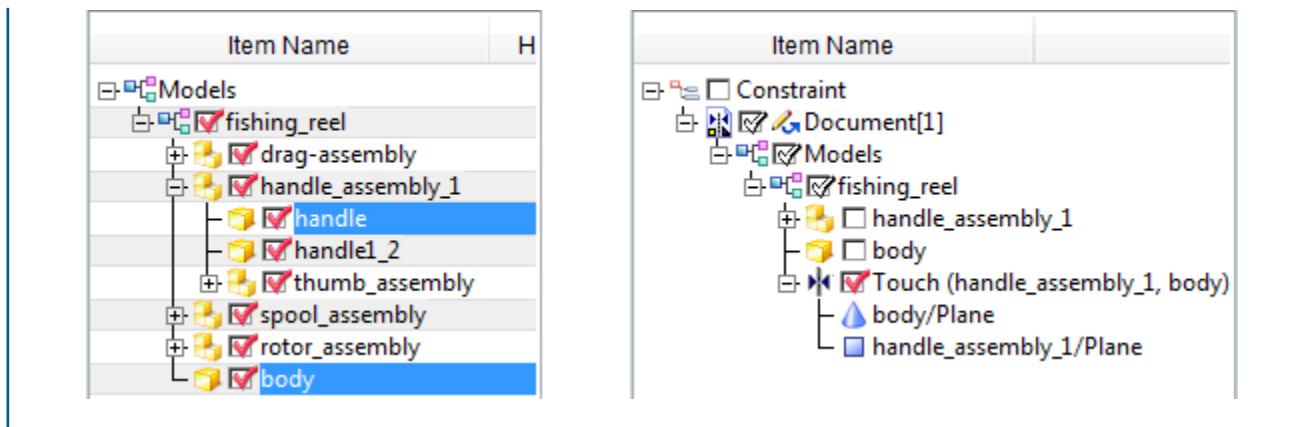
Tip:

If **Show Leaf Structure** is activated, when you select a component in the Viewing window, the leaf node is selected, not the component. To select the component, either select it from the assembly tree or deactivate **Show Leaf Structure**.



Example:

A constraint is created between the handle and the body. The nearest common owner is fishing_reel. The constraint is created between handle_assembly_1 (node immediately below the nearest common owner) and body, and it is placed at fishing_reel.



Simulating mechanisms using constraints

To simulate motion between parts or subassemblies that are in different subassemblies, you can use the mechanism feature. A mechanism consists of multiple components, possibly from different subassemblies, linked together.

To simulate the movements of the mechanism with the constraint system, all the moveable mechanism components must be identified as mechanism components using **Create Mechanism Link**. All the constraints on mechanism components are created at the root node instead of the nearest common owner.

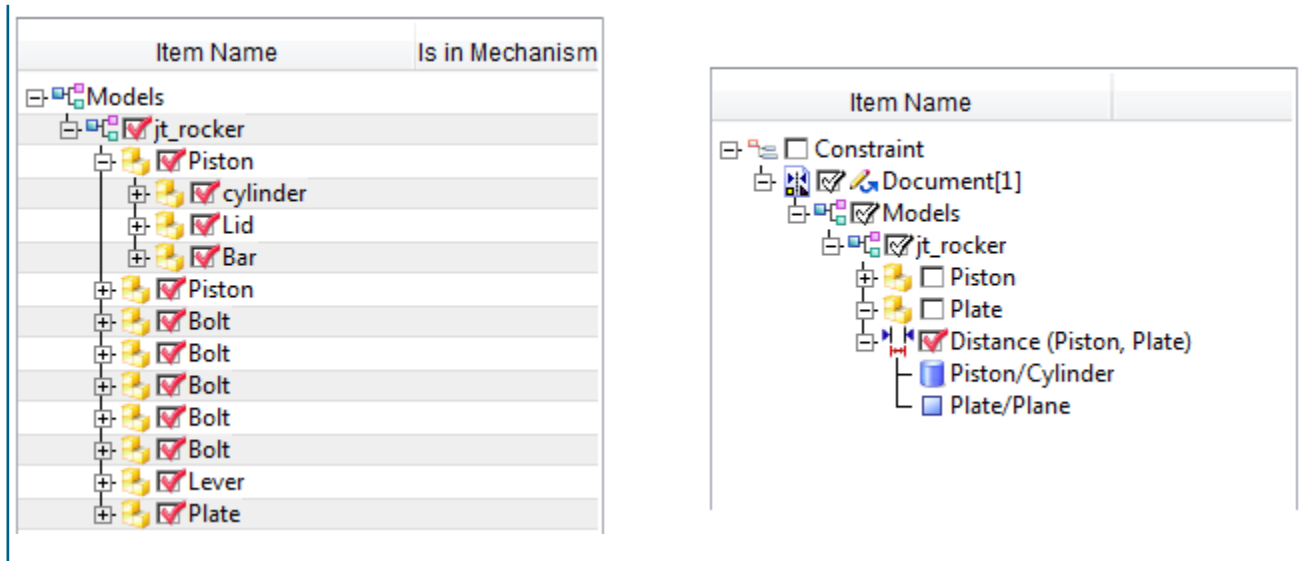
When you create a constraint between components from different subassemblies without a mechanism, the constraint is created on the nearest common parent object.

Tip:

If **Show Leaf Structure** is activated, when you select a component in the Viewing window, the leaf node is selected, not the component. To select the component, either select it from the assembly tree or deactivate **Show Leaf Structure**.

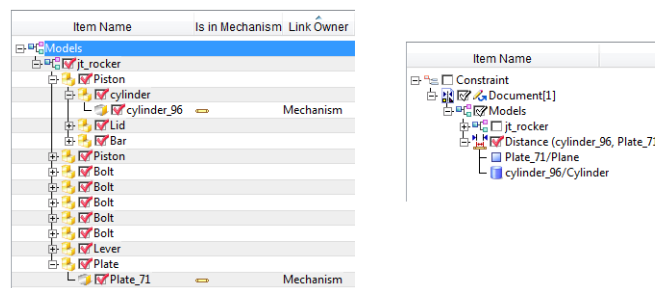
Example:

A constraint between the cylinder and the plate is created on the jt_rocker node.



Example:

When you create a mechanism link between the components, the constraint is created at the root of the structure.



Create mechanisms to constrain components

Mechanisms may have components from different subassemblies. To simulate the movement of the mechanism using the constraint system, you create a mechanism link for each component you want to add to the mechanism. A constraint on a mechanism link is created between selected components (part/sub-assembly/leaf node) and placed at the root node, whereas a constraint between subassemblies (assembly positioning) is placed at the nearest common owner.

Mechanisms are not necessary for constraining moving parts and subassemblies that belong to same parent component.

If you already have a constraint between components for which you want to create a mechanism, you can **create a mechanism for an existing constraint**.

Note:

If you create a constraint between a mechanism component and a non-mechanism component, the constraint is created at the root of the structure, and any pre-existing constraints on the non-mechanism component are copied to the root.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. From the **3D Constraints** toolbar, click **New Constraint File** .
3. For each component to add to the mechanism:
 - a. Select the component in the BOM structure.
 - b. Right-click **Document** in the **Constraint** tree, and choose **Create Mechanism Link**.

The mechanism symbol appears in the **Is In Mechanism** column.

4. **Create the constraint** between the mechanism components.

The constraint is created at the root of the structure. The constraint label appears near the components and includes a mechanism symbol.

Create a mechanism for an existing constraint

To use the mechanism feature to constrain components that are part of two different subassemblies, you create a mechanism link for each component you want to add to the mechanism and then create the constraint. If you already have a constraint between components without a mechanism, the constraint exists at the components' nearest common owner. You can create a mechanism for the existing constraint.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. From the **3D Constraints** toolbar, click **New Constraint File** .
3. For each component to add to the mechanism:
 - a. Select the component in the BOM structure.
 - b. Right-click **Document** in the **Constraint** tree, and choose **Create Mechanism Link**.

The other component referenced by the constraint is automatically converted to a mechanism link.

The original constraint created between the components at the nearest common owner becomes hidden and is copied to the root of the model.

The mechanism symbol appears in the **Is In Mechanism** column.

4. Select one of the components in the BOM structure, and then in the **Constraint** tree, right-click **Document**, and choose **Remove Mechanism Link**.

The constraint that was copied to the root of the model is removed, and the constraint at the nearest common owner is activated.

Move components by constraints

You can move a group of components together by using the **Components to Move** option in the **Create Constraints** dialog box. When you use this option, a bond constraint and mechanisms are automatically created for the components you choose to group. You can create a constraint between this group and another component.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. If you do not have an open constraint file, do one of the following:

To	
Create a constraint file	From the 3D Constraints toolbar, click New Constraint File  .
Open an existing constraint file	From the 3D Constraints toolbar, click Open Constraint File  .

3. From the **3D Constraints** toolbar, click **Create Constraint** .
4. In the **Create Constraints** dialog box, select either **Associative** or **Positional**.
5. Select **Components to Move**.
6. Click **Select**.
7. Select the components you want to group together.
8. Click **Done**.
9. Create the constraint between the parts you grouped together and another component:
 - a. Select one of the components you grouped and a new component.

b. From the **Constraint Type** list, choose one of the following:

- **Touch**
- **Distance**
- **Parallel**
- **Perpendicular**
- **Angle**

c. **Create the constraint**, following the steps for the specific constraint type.

If necessary, click **Highlight Previous** to see view the previously selected components used to create the constraint.

A bond constraint and mechanisms are created for the components you grouped.

The constraint is created at the root of the assembly.

Set constraint preferences

Adjust constraints preferences to change the behavior of constraints and the appearance of new constraint labels.

Note:

Changing constraint label appearance preferences affects new constraints. To change the appearance properties of an existing label, right-click a label and choose **Properties**.

1. From the **3D Constraints** toolbar, click **Constraint** .
2. On the **3D Constraints** toolbar, click **Preferences** .
3. In the **Constraint Preferences** dialog box, change the preferences in one of these ways:

On this tab	Use this option	To
General	Create new Constraint document	Create a constraint document when you load a constraint file. If you already have a constraint

On this tab	Use this option	To
		document, your constraints are deleted and replaced by the constraints in the file.
	Merge Constraints into existing document	Add the constraints in a saved constraint file to your constraint document. If you do not have a constraint document, create one.
	Suppress loaded Constraints	Automatically suppress all constraints that you load from a file. This option is only available when the Merge constraints into existing document preference is selected.
	Honor Constraints	Specify whether or not to solve constraints and reposition all affected parts as you create constraints, play motion files, or transform parts.
	Move fixed components	Allow components with Fixed constraints to move when you play motion files or transform parts.
	Show Constraint Annotations	Specify whether annotations for constraints appear in the Viewing window. Hiding them improves performance if playing motion files or transforming parts is slow.
Line	Line style	Specify the style of constraint label leader lines.
	Line color	Specify the color of constraint label leader lines.
	Thickness	Specify the thickness, in points, of constraint label leader lines.
	Display leader lines on top	Always display constraint label leader lines on top of models, even when they are behind geometry.
Edge	Edge style	Specify the style of the borders around constraint labels.
	Edge color	Specify the color of the borders around constraint labels.
	Thickness	Specify the thickness, in points, of the borders around constraint labels.
Fill	Fill style	Specify the style of the fill of constraint labels.
	Fill color	Specify the color of the fill of constraint labels.

On this tab	Use this option	To
	Transparency	Specify the level of transparency (from Transparent to Opaque) of the fill of constraint labels.
Font	Font	Specify the font used in constraint labels.
	Font Style	Specify the style of the font used in constraint labels.
	Size	Specify the size of the font used in constraint labels.
	Underline or Strikeout	Specify special font effects used in constraint labels.
	Color	Specify the color of the font used in constraint labels.
Highlight	Highlight Color	Specify the selection color of the selected part features.
	Thickness	Specify the thickness, in pixels, of the feature selection highlight. You can specify a thickness value of 1-10.
	Suppressed Color	Specify the color of constraint label leader lines for suppressed constraints.
	Out Of Date Color	Specify the color of constraint label leader lines for constraints that are out of date.
	Error Color	Specify the color of constraint label leader lines for constraints with errors.
	Display Status on Annotations	Specify whether to display status along with constraint type in the constraint label. 
	Update Annotation on Still Frames Only	Specify whether to move constraint labels as the parts move.

- Click **OK**.

How constraints affect motion clients

In Mockup, you can specify how constraints affect most motion clients on the **General** tab of the **Constraints Preferences** dialog box.

Use this option	To
Honor Constraints	Specify whether or not to solve constraints and reposition all affected parts as you create constraints, play motion files, or transform parts.
Move fixed components	Allow components with Fixed constraints to move when you play motion files or transform parts.
Show Constraint Annotations	Specify whether annotations for constraints appear in the Viewing window. Hiding them improves performance if playing motion files or transforming parts is slow.

Constraints are honored by the following motion clients (in Mockup the **Honor Constraints** preference must be set):

- **Motion file playback**
- **Reset All Parts**
- **Part Transformation dialog box**
- **Part manipulators**
- **Part manipulation mode**
- **Relocate parts**

Constraints are always ignored by the following motion clients:

- **Explode view**
- **Thrustlines**
- **Path planning**

Loading constraints from a file

Load a constraint file

1. From the **3D Constraints** toolbar, click **Constraint** .
2. From the **3D Constraints** toolbar, click **Open Constraint File** .

3. In the **Select Constraint Files** dialog box, select the file and click **Open**.

Constraint labels appear in the Viewing window and the constraint document appears in the Constraints view.

4. (Optional) To apply the constraints to the assembly, do one of the following:
 - Choose **Constraint→Update**.
 - In the Constraints view, right-click the constraint document and choose **Update**.

Merge constraint files

You can merge constraints from a constraint file into your active constraint document.

1. If you do not have an open constraint file, do one of the following:
 - Open a constraint file.
 - Create a new constraint file.
2. On the **3D Constraints** toolbar, click **Preferences** .
3. On the **General** tab of the **Constraints Preferences** dialog box, select the **Merge Constraints into existing document** check box and click **OK**.
4. From the **3D Constraints** toolbar, click **Open Constraint File** .
5. In the **Select Constraint Files** dialog box, select the file and click **Open**.

Constraint labels appear in the Viewing window and the constraints appear in the Constraints tree.

6. (Optional) To apply the constraints to the assembly, do one of the following:
 - Choose **Constraint→Update**.
 - In the Constraints view, right-click the constraint document and choose **Update**.

Saving constraints

Overview of saving constraints

When you create constraints, they are added to the active constraint document. If you save the constraints, you can apply the constraints again later. There are several ways to save constraints. You can:

- Save constraints to a constraint file.
- Save constraints as part of a session file.
- Save constraints as part of a PLM XML file.
- Save constraints as a Teamcenter *DirectModel Constraint* dataset.

When you save constraints as part of a session file, the session file also includes the properties and positions of constraint labels.

Save constraints to a constraint file

1. Create a new constraint file or open an existing file.
2. Add or modify constraints.
3. Do one of the following:

To	Do this
Save a new constraint file	a. Do one of the following: • On the 3D Constraints toolbar, click Save Constraint File . • In the Constraints view, right click the document node and choose Save Constraint File. b. In the Select Constraint Files dialog box, specify the path and filename, and click Save.
Save an existing constraint file to the same name	Do one of the following: • On the 3D Constraints toolbar, click Save Constraint File .

To	Do this
	In the Constraints view, right click the document node and choose Save Constraint File.
Save an existing constraint file to a new file	- Do one of the following: - On the 3D Constraints toolbar, click Save As Constraint File . - In the Constraints view, right click the document node and choose Save As Constraint File. - In the Select Constraint Files dialog box, specify the new path and filename, and click Save.

Managing constraints

Change constraint distance or angle value

Change the distance or angle of Distance and Angle constraints.

- Do one of the following:
 - In the Viewing window, right-click a constraint label and choose **Redefine**.
 - In the Constraints tree, right-click the constraint name and choose **Redefine**.
- In the **Constraint Edit** dialog box, type the new distance or angle and click **Apply**.

One of the components moves as the constraint is resolved, and the constraint label shows the new value.

Overview of constraints in the Constraints view

You can view and manage constraints in the Constraints view. Constraints appear in the Constraints tree. You can choose to display status information in columns.

Item Name	Status	Status Information	Type	Value
Constraints				
Document[1]	✓	Solved		
Models				
1_PMI_Block				
1_Pin	○	Partially Constrained		
2_Pin	○	Partially Constrained		
5_Block	●	Fully Constrained		
6_Fixture	✓	Fixed		
Angle	✓	Solved		35.000000
1_Pin/Cylinder : (Axis)	○	Under Defined		
2_Pin/Cylinder : (Axis)	○	Under Defined		
Distance	✓	Solved		35.000000
Distance	✓	Solved		15.000000
5_Block/Plane	●	Well Defined		
6_Fixture/Plane	✓	Fixed		
Fixed	✓	Fixed		
Parallel	✓	Solved		
Perpendicular	✓	Solved		
Touch	✓	Solved		
5_Block/Plane	●	Well Defined		
6_Fixture/Plane	✓	Fixed		

The tree contains the following types of nodes:

- Constraints (the root node)
- Constraint document
- Models
- Subassemblies
- Components
- Constraints
- Geometry references

Manage constraints in the Constraints tree

You can do any of the following in the Constraints view:

To	Right-click	And choose
Load an existing constraint file	Constraints (the root node)	Open Constraint File
Create a new constraint file		New Constraint File
Set constraint preferences		Preferences
Suppress or unsuppress all constraints	The constraint document	Suppress Constraints
Solve all constraints		Update
Expand the entire Constraint tree		Expand
Save a constraint file		Save Constraint File
Save a constraint file with a new name		Save As Constraint File
Delete all constraints		Delete
Create a constraint		Create Constraint
Show or hide a component's degrees of freedom	A constrained component	Show Degrees Of Freedom
Change a distance or angle value	A constraint	Redefine
Rename a constraint		Rename
Show another solution		Reverse
Show other solutions		Cycle
Suppress a constraint		Suppress
Delete a constraint		Delete

Understanding constraint status information

In the Constraints view, if the **Status** and **Status Information** columns are displayed, you can see the status of the constraint document, components, constraints, and geometry references in the Constraint tree.

The following table describes the status of constraint documents:

This icon	With this status information	Indicates
	Solved	All constraints are solved and all components are positioned correctly.
	Not Solved	Constraints have been modified and the system needs to be solved.

This icon	With this status information	Indicates
	Not Consistent	Constraints are defined such that the system is not consistent.
	Over Constrained	Constraints are defined such that the system is over constrained.
	Unknown Failure	There was an unknown error during solve.
	UnResolved Constraints	The document contains unresolved constraints.

The following table describes the status of components:

This icon	With this status information	Indicates
	Fully Constrained	Component is fully constrained.
	Partially Constrained	Component is partially constrained.
		Component is unconstrained or the component is suppressed
		Component is inconsistently constrained.
	Fixed	Component is fixed.

The following table describes the status of constraints:

This icon	With this status information	Indicates
	Solved	Constraint is solved and components are positioned correctly.
	Over Defined	Conflicts with another constraint.
		The objects referenced by this constraint cannot move relative to one another.
		No attempt could be made to solve this constraint due to conflicts between other constraints. You should examine and fix problems with other constraints to allow this constraint to solve.
		Invalid geometry type for this constraint.

This icon	With this status information	Indicates
		This constraint cannot solve. Either it references a component that has been deleted or suppressed, or it references geometry that has been deleted or suppressed.
		Could not solve as some geometry is unloaded or missing.
		Newly created and not yet solved. This could be because the Honor Constraints preference is off.
		Document ignores all constraints.
	UnResolved	The constraint contains unresolved geometry references.

The following table describes the status of geometry references:

This icon	With this status information	Indicates
	Unknown	Unknown status (newly added to system).
	Fixed	Geometry has been declared fixed.
	Over Defined	Geometry is over defined by dimensions.
	Not Consistent	Geometry inconsistently defined by dimensions.
	Not Consistent – Dimensions	Geometry inconsistently defined by dimensions – change model.
	Not Consistent – Other	Geometry inconsistently defined by dimensions – try changing solving mode.
	Not Consistent – Unknown	Geometry inconsistently defined by dimensions – cannot ascertain what to recommend.
	Not Changed	Geometry was not changed.
	Well Defined	Geometry is well defined by dimensions.
	Under Defined	Geometry is under defined by dimensions.
	Under Defined – Bounded Dimension	Geometry is under defined but only due to a bounded dimension.
	UnResolved	Geometry reference is unresolved.

Display and hide columns in the Constraints view

The Constraints tree appears in the **Item Name** column in the Constraint view. You can display and hide additional columns.

1. Right-click a column heading.
2. In the column list, select or clear any of the following:
 - **Status**
 - **Status Information**
 - **Type**
 - **Value**
 - **File Name**
 - **File Location**
3. (Optional) Drag column headings to reorder and resize the columns.

Suppress constraints

Suppress constraints to remove them from consideration without deleting them.

To suppress or unsuppress constraints do one of the following:

To suppress or unsuppress	Do this
All constraints	In the Constraints tree, right-click the constraint document and choose Suppress Constraints. Note: Using Suppress Constraints does not unsuppress constraints that were suppressed individually.
Individual constraints	Do one of the following: • In the Viewing window, right-click a constraint label and choose Suppress. • In the Constraints tree, right-click the constraint and choose Suppress.

To suppress or unsuppress	Do this
	Note: You cannot unsuppress individual constraints if you used Suppress Constraints to suppress all constraints.

In the Constraint tree, suppressed constraints appear in italics. In the Viewing window, by default, suppressed constraint label leader lines appear as dashed lines.

Delete constraints

Do one of the following:

To delete	Do this
Individual constraints	Do one of the following: - In the Viewing window, right-click a constraint label and choose Delete. - In the Constraints tree, right-click the constraint and choose Delete.
All constraints	In the Constraints tree, right-click the constraint document and choose Delete .

Note:

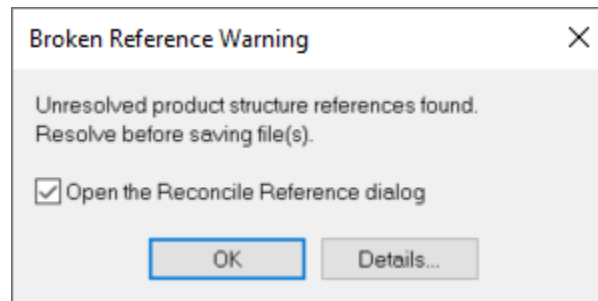
You can use the **Undo** command to undo the deletion.

Reconcile broken constraint part references

As you change or re-translate assembly models, you may break constraint file part references. For example, if you constrain parts in an assembly and save the constraint file, and then later you replace a part in the assembly with a new part, all references to the old part are broken.

If you reuse a constraint file with a different model, you will see broken part references. For example, you constrain an assembly model and save its constraint file, and then you open a similar model and load the first constraint file. The constraint system reports broken references.

You can reconcile broken references by creating new mappings. If the **Broken Reference Warning** dialog box appears when you load a constraint file or open a session file that includes constraints, do the following to reconcile the broken part references:



1. In the **Broken Reference Warning** dialog box, select the **Open the Reconcile Reference dialog** check box and click **OK**.
2. In the **Reconcile Reference Dialog** dialog box, select a broken reference in the tree.
3. Do one of the following:
 - Select the reconcile candidate from the candidate list.
 - Use Find to search for a candidate, and then select it.
 - In the Viewing window or the assembly tree, select a part or assembly.
 - If the broken reference is not needed, and you want to permanently delete the reference from the tree and the session, click **Ignore**.
4. Click **Update**.

The reference to the part or assembly is updated in the **Reconcile Reference** dialog box.

5. Repeat the steps above to create a new mapping for each broken reference.
6. Click **Close**.
7. On the **Reconcile** toolbar, click **Reload Session** .
8. If the broken reference were found when you loaded a constraint file, load the constraint file again.
9. (Optional) Export a mapping file, and use it to automatically resolve broken references in other files that reference the same data.

Resolve unresolved constraint geometry references

If the referenced part changes, the constraint system will not know what geometry to reference on the changed part. You can redefine the constraints to reference the correct geometry. The process is very similar to creating constraints.

1. In the Constraints view, expand Constraints, and then right-click the constraint document node and choose **Expand** to expand the entire tree.

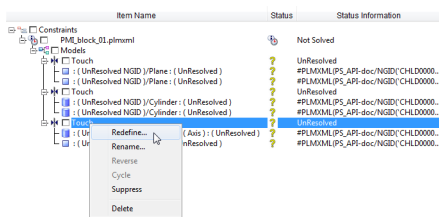
The entire Constraints tree expands.

2. Locate the first unresolved reference to resolve.

Tip:

Right-click the header area of the Constraints view and choose **Status** and **Status Information** to see the status of each constraint.

3. In the tree, right-click an unresolved constraint and choose **Redefine**.



4. In the **Redefine Constraints** dialog box, recreate the constraint.
 - Depending on the issue, you may need to select geometry on only one of the components.
 - For **Distance** and **Angle** constraints, retype the distance or angle value.
5. Click **Apply**.

The constraint label appears in the Viewing window and the constraint status updates in the Constraints tree.

6. Repeat the process for the remaining unresolved constraints.

Working with constraint labels

Overview of constraint labels

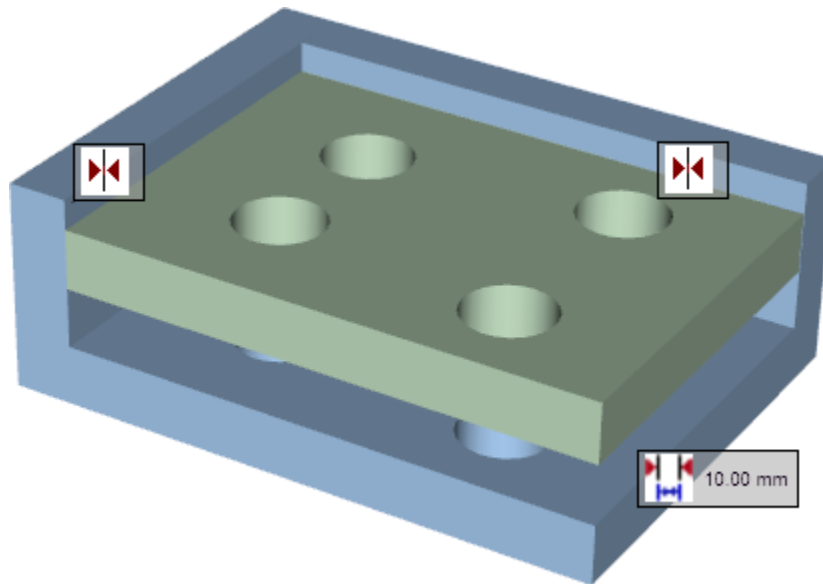
Constraint labels appear next to constrained components. The label shows the constraint type. Distance and Angle constraints also show the distance or angle value.

You can:

- Manipulate constraint labels.

- Show and hide constraint labels.
- Set a preference to include the constraint status in the label.
- Set preferences to control the appearance of constraint labels and leader lines.

When you save constraints as part of a session file, the session file also includes the properties and positions of constraint labels.



Manipulate constraint labels

Do any of the following:

To	Do this
Hide a constraint label.	Do any of the following: • In the Viewing window, right-click a constraint label and choose Hide Constraint. • In the Constraints view, clear the check box for the constraint.
Show a hidden constraint label.	In the Constraints view, select the check box for the constraint.
Reposition a constraint label.	Drag the constraint label in the Viewing window.
Change the appearance properties of a constraint label and its leader lines.	1. In the Viewing window, right-click the constraint label and choose Properties .

To	Do this
	2. Change the preferences in the Constraint Preferences dialog box.

Find a constraint in the Constraint tree

Right-click a constraint label and choose **Show in Tree**.

If necessary, the Constraint tree expands to show the constraint. The constraint is selected in the tree.

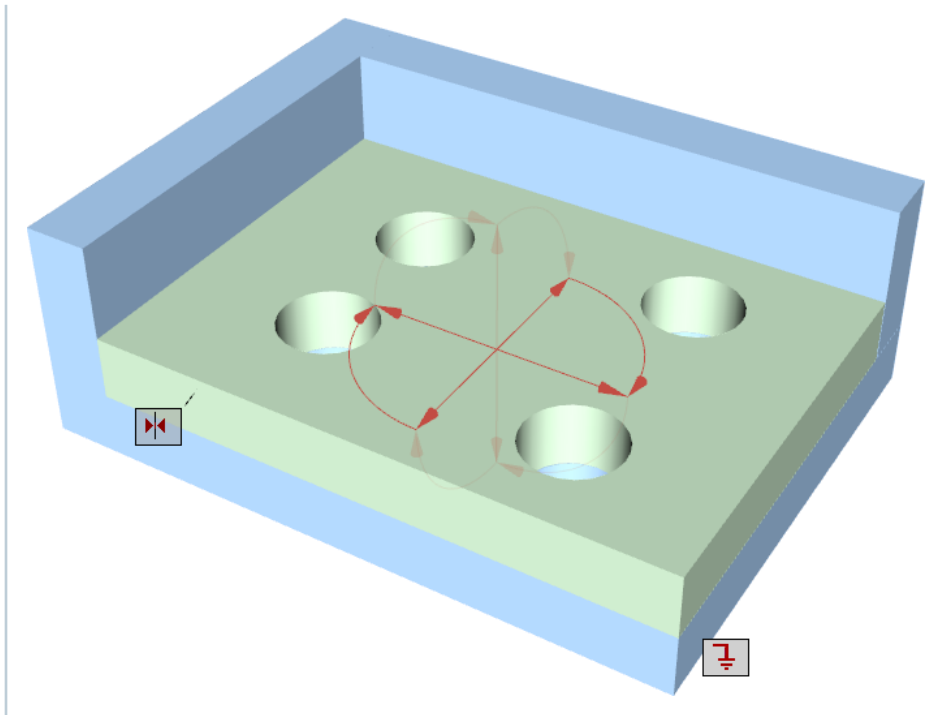
Viewing Degrees of Freedom

Degrees of Freedom overview

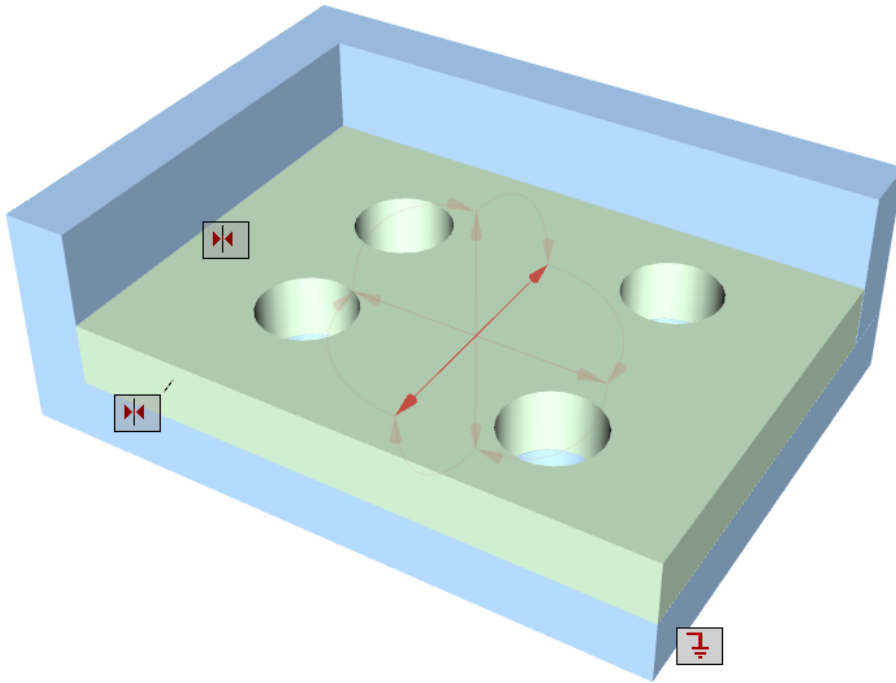
An unconstrained component has six degrees of freedom. You can transform it in the X, Y, or Z direction and rotate it around the X, Y, or Z axis. A component with a Fix constraint cannot move. It has zero degrees of freedom. When you add constraints between components, some or all of the degrees of freedom are lost.

You can display the degrees of freedom for selected components with constraints. The degrees of freedom appear in the Viewing window as dark red arrows. Directions with no degrees of freedom are shown as light red arrows.

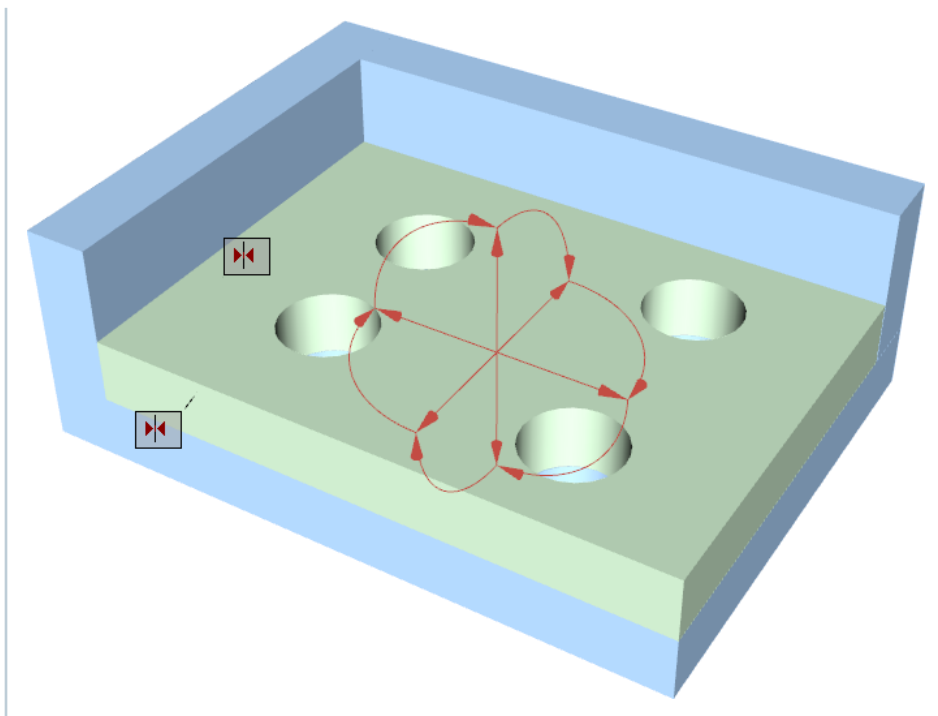
In this example, the light blue part is fixed. A Touch constraint was added between the bottom of the light green part and the top of the light blue part. You can see the degrees of freedom of the light green part. It cannot move up and down. It can rotate around only one axis.



For this example, a second Touch constraint was added between the left end of the green part and the inside face of the blue part. You can see the degrees of freedom of the green part. It can move in only one direction. It cannot rotate.



In this example, the blue part is no longer fixed. You can see the degrees of freedom of the green part. It is free to move and rotate (along with the blue part) in all directions. Both parts have six degrees of freedom.



Show and hide Degrees of Freedom

You can show the Degrees of Freedom for selected components with constraints.

Do any of the following:

To	Do this
Show degrees of freedom for a part	Do one of the following: - In the Viewing window, right-click a component with a constraint and choose Degrees of Freedom. - In the Constraints tree, right-click the component and choose Show Degrees of Freedom.
Hide degrees of freedom for a single part	Do one of the following: - In the Viewing window, right-click a component with a constraint and choose Degrees of Freedom. - In the Constraints tree, right-click the component and choose Hide Degrees of Freedom.
Hide all visible degrees of freedom	In the Viewing window, right-click the background and choose Clear Degrees of Freedom .

Aligning parts using positional constraints

Using positional constraints to align components

If your models contain B-Rep data, but not Entity IDs, you cannot create associative constraints, but you can still position components in an assembly using positional constraints. You cannot save positional constraints.

You can use positional constraints to:

- Align components so they touch or align with each other.
- Specify a distance or angle between aligned components.
- Specify that a component is fixed in place so it does not move when you align other components to it.

Constraint type	Description
 Touch	Aligns two components such that the selected two geometric features are tangent or coincident depending on the geometry.
 Distance	Specifies the distance between two components.
 Fix	Fixes a component so it does not move as you align other components.
 Parallel	Aligns two components such that the selected two geometric features are parallel.
 Perpendicular	Aligns two components such that the selected two geometric features are perpendicular.
 Angle	Aligns two components such that the selected two geometric features are at a specified angle.
 Bond	Aligns two or more components so they move as a single rigid unit.

Workflow for positioning components using positional constraints

The general workflow for positioning components using positional constraints is nearly identical to the workflow for creating associative constraints.

1. In the **Create Constraints** dialog box, select **Positional**.
2. Specify the constraint type.
3. Select the components to fix or component geometry to constrain.

4. Depending on the type of geometry selected, refine the selection.
5. For Distance and Angle constraints, specify the distance or angle value.
6. If necessary, click **Reverse** or **Cycle** to see other possible positioning.

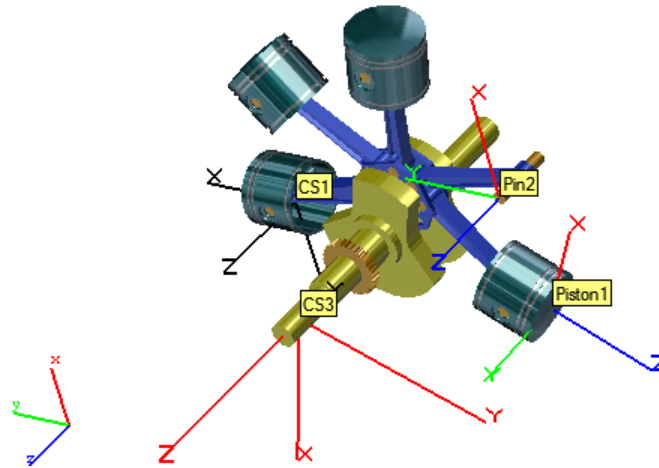
Note:

You can also use 3D **Alignment** tools to align parts in an assembly.

7. Working with coordinate systems

Overview of coordinate systems

There are three types of coordinate systems you can work with: Global, PMI, and User Defined.



Global coordinate system

When you open a 3D model, you are working in the Global coordinate system. If it is displayed, the **3D trihedron** in the lower left corner of the Viewing window shows you the orientation of the Global coordinate system. The Global coordinates system is used for measuring and transforming parts if you do not designate a Working Coordinate System.

PMI coordinate systems

Some models contain PMI coordinate systems. PMI coordinate systems are defined in the CAD system.

You can:

- Manage PMI coordinate systems.
- Choose a PMI coordinate system to be the Working Coordinate System (Working CS) and use it for **measuring** and **transforming** parts.
- Align your view to a PMI coordinate system.
- Set appearance preferences for PMI coordinate systems.
- Select a PMI coordinate system origin, endpoint, or axis for tasks that require selecting part features.

User Defined coordinate systems

You can create new coordinate systems. User Defined coordinate systems are saved when you save a session file.

You can:

- Create User Defined coordinate systems.
- Manage User Defined coordinate systems.
- Manipulate User Defined coordinate systems.
- Edit User Defined coordinate systems.
- Choose a User Defined coordinate system to be the Working Coordinate System (Working CS) and use it for **measuring** and **transforming** parts.
- Delete User Defined coordinate systems.
- Align a part or assembly to a User Defined coordinate system.
- Align your view to a User Defined coordinate system.
- Set appearance preferences for User Defined coordinate systems.
- Select a User Defined coordinate system origin, endpoint, or axis for tasks that require selecting part features.

Create coordinate systems

Create Coordinate System

Name

Csys1

Align to Feature

Align Only:

Pick Feature

Align and Associate:

Pick Feature

Part Association:

Remove

Position

X:

0.000000

Y:

0.000000

Z:

0.000000

Orientation (degrees)

I:

0.000000

J:

0.000000

K:

0.000000

OK

Cancel

Apply

1. On the **3D Coordinate System** toolbar, click **Create Coordinate System** .

The new coordinate system and a manipulator appear at the origin of the Global coordinate system. If the **Show label** preference is set, the default name appears next to the new coordinate system.

2. (Optional) Use the manipulator to reposition the coordinate system.
3. (Optional) In the **Create Coordinate Systems** dialog box do any of the following:

To	Do this
Rename the coordinate system	In the Name section, type a new name.
Align the coordinate system with a part feature	In the Align to Feature section, next to Align Only , click Pick Feature , and then click a part feature in the Viewing window. The coordinate system moves to align with the part feature.
Align the coordinate system with a part feature and	In the Align to Feature section, next to Align and Associate , click Pick Feature , and then click a feature on the part.

To	Do this
associate the coordinate system with the part	The coordinate system moves to align with the part feature, and the name of the associated part appears in the Part Association section. If you transform the part, the coordinate system will transform with it.
Specify coordinates of the origin	In the Position section, type X, Y, Z values.
Specify the orientation of the axes	In the Orientation section, type I, J, K values.

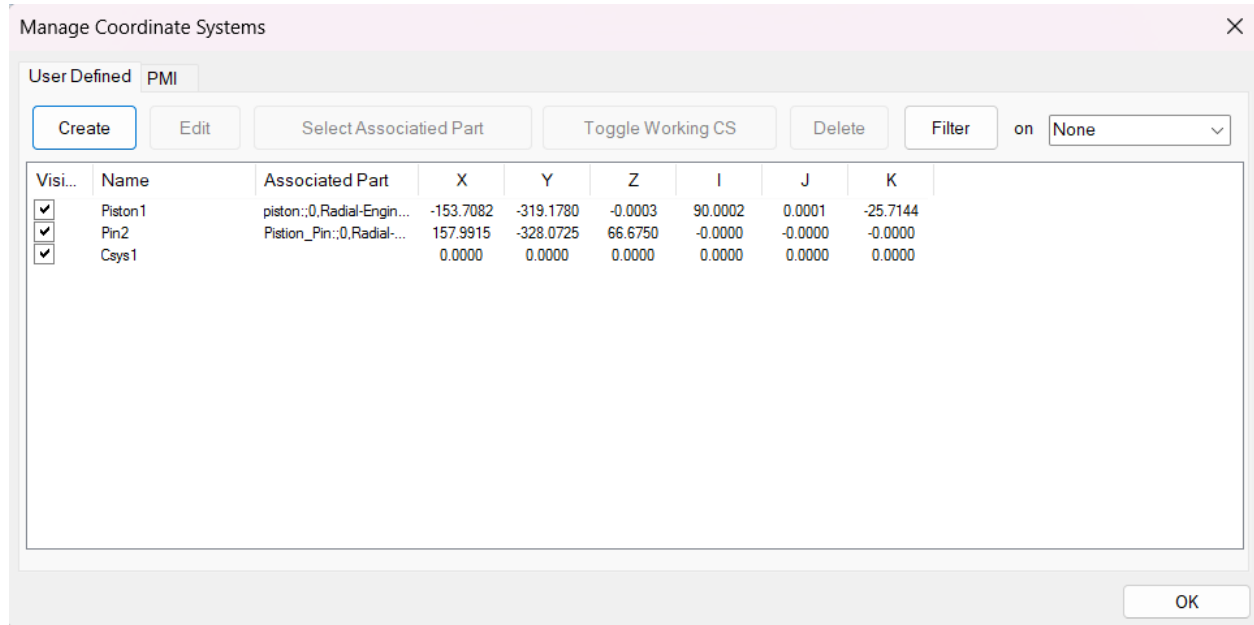
4. Do one of the following:

To	Do this
Discard the new coordinate system before applying changes	Click Cancel .
Apply the changes without closing the dialog box	Click Apply .
To accept the new coordinate system and close the dialog box	Click OK . The manipulator disappears but the coordinate system trihedron remains.

Note:

If you cannot see the new coordinate system, it may be hidden by parts of the model. Try zooming out, rotating your view, or changing the trihedron size in the **Coordinate System Preferences** dialog box.

Manage coordinate systems



- Do any of the following:
 - In the Viewing window, right-click a coordinate system and choose **Manage**.
 - On the **3D Coordinate System** toolbar, click **Manage Coordinate Systems** .

The **Manage Coordinate Systems** dialog box appears. It has two tabs, **User Defined** and **PMI**.

- (Optional) To manage user-defined coordinate systems, click the **User Defined** tab and choose any of the following options:

Use this option	To
Create.	Create a new coordinate system.
Edit	Edit the selected coordinate system.
Select Associated Part	Select the part that is associated with the selected coordinate system. The option is unavailable if no part is associated with the selected coordinate system.
Toggle Working CS	(If it is not already the Working Coordinate System) Designate the selected coordinate system as the Working Coordinate System. (If it is already the Working Coordinate System) Turn off the Working Coordinate System designation.
Delete	Delete the selected coordinate system.

Use this option	To												
Filter	Specify which coordinate systems to show in the list when you open the Manage Coordinate Systems dialog box. Select a filter from the Filter list, and then click Filter. <table> <tr> <th>Choose</th><th>To</th></tr> <tr> <td>None</td><td>Cancel filtering and list all coordinate systems.</td></tr> <tr> <td>Selected Parts</td><td>List only filters associated with the selected parts.</td></tr> <tr> <td>Visible Parts</td><td>List only filters associated with visible parts.</td></tr> <tr> <td>Selected Csys</td><td>List only the selected coordinate system.</td></tr> <tr> <td>Visible Csys</td><td>List only visible coordinate systems.</td></tr> </table>	Choose	To	None	Cancel filtering and list all coordinate systems.	Selected Parts	List only filters associated with the selected parts.	Visible Parts	List only filters associated with visible parts.	Selected Csys	List only the selected coordinate system.	Visible Csys	List only visible coordinate systems.
Choose	To												
None	Cancel filtering and list all coordinate systems.												
Selected Parts	List only filters associated with the selected parts.												
Visible Parts	List only filters associated with visible parts.												
Selected Csys	List only the selected coordinate system.												
Visible Csys	List only visible coordinate systems.												
Check boxes in the Visible column	Turn coordinate systems on and off.												
Column headers	Sort the list on the selected column.												

3. (Optional) To manage PMI coordinate systems, click the **PMI** tab and choose any of the following options:

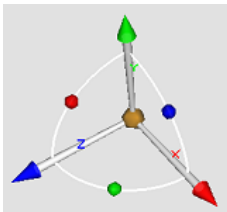
Use this option	To								
Load (from)	Add coordinate systems to the list by loading PMI coordinates systems associated with the specified parts. Select the parts from the Load from list, and then click Load. <table> <tr> <th>Choose</th><th>To</th></tr> <tr> <td>All Parts</td><td>Load all PMI coordinate systems.</td></tr> <tr> <td>Selected Parts</td><td>Load only the PMI coordinate systems associated with the selected parts.</td></tr> <tr> <td>Visible Parts</td><td>Load only the PMI coordinate systems associated with the visible parts.</td></tr> </table>	Choose	To	All Parts	Load all PMI coordinate systems.	Selected Parts	Load only the PMI coordinate systems associated with the selected parts.	Visible Parts	Load only the PMI coordinate systems associated with the visible parts.
Choose	To								
All Parts	Load all PMI coordinate systems.								
Selected Parts	Load only the PMI coordinate systems associated with the selected parts.								
Visible Parts	Load only the PMI coordinate systems associated with the visible parts.								
Select Associated Parts	Select the parts that are associated with the selected coordinate system.								
Toggle Working CS	(If it is not already the Working Coordinate System) Designate the selected coordinate system as the Working Coordinate System. (If it is already the Working Coordinate System) Turn off the Working Coordinate System designation.								

Use this option	To												
Filter	Specify which coordinate systems to show in the list when you open the Manage Coordinate Systems dialog box. Select a filter from the Filter on list, and then click Filter. <table> <tr> <th>Select</th><th>To</th></tr> <tr> <td>None</td><td>Cancel filtering and list all coordinate systems.</td></tr> <tr> <td>Selected Parts</td><td>List only filters associated with the selected parts.</td></tr> <tr> <td>Visible Parts</td><td>List only filters associated with visible parts.</td></tr> <tr> <td>Selected Csys</td><td>List only the selected coordinate system.</td></tr> <tr> <td>Visible Csys</td><td>List only visible coordinate systems.</td></tr> </table>	Select	To	None	Cancel filtering and list all coordinate systems.	Selected Parts	List only filters associated with the selected parts.	Visible Parts	List only filters associated with visible parts.	Selected Csys	List only the selected coordinate system.	Visible Csys	List only visible coordinate systems.
Select	To												
None	Cancel filtering and list all coordinate systems.												
Selected Parts	List only filters associated with the selected parts.												
Visible Parts	List only filters associated with visible parts.												
Selected Csys	List only the selected coordinate system.												
Visible Csys	List only visible coordinate systems.												
Check boxes in the Visible column	Turn coordinate systems on and off.												
Column headers	Sort the list on in the selected column.												

Manipulate coordinate systems

- Do one of the following to display a coordinate system manipulator:
 - In the Viewing window, right-click a coordinate system and choose **Manipulate**.
 - Select the coordinate system, and then on the **3D Coordinate System** toolbar, click **Manipulate Selected Coordinate System** .
 - Create a coordinate system.
 - Edit a coordinate system.

A manipulator appears around the coordinate system. The manipulator is an interactive triad with three axis arrows for translating and three small spheres for rotating.



- (Optional) Repeat the previous step to add manipulators to other coordinate systems.

3. If you have more than one manipulator, click a manipulator to activate it.

The active manipulator has brighter colors. Only one manipulator is active at any time.

4. Transform the coordinate system in any of these ways:

To	Do this						
Translate along X, Y, or Z	Drag the X, Y, or Z manipulator axis. (Red is X, green is Y, and blue is Z.)						
Translate precise distances in X, Y, or Z	a. If the Position window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click the X, Y, or Z manipulator axis. c. Do one of the following: <table border="1"> <thead> <tr> <th>To</th><th>Do this</th></tr> </thead> <tbody> <tr> <td>Drag using a snap increment</td><td> A. (Optional) Type a new Snap value in the Position window. B. Drag the axis. If Snap is set to 5, the coordinate system moves in 5 unit increments. </td></tr> <tr> <td>Specify a delta distance</td><td> Enter the Distance in the Position window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value. </td></tr> </tbody> </table>	To	Do this	Drag using a snap increment	A. (Optional) Type a new Snap value in the Position window. B. Drag the axis. If Snap is set to 5, the coordinate system moves in 5 unit increments.	Specify a delta distance	Enter the Distance in the Position window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.
To	Do this						
Drag using a snap increment	A. (Optional) Type a new Snap value in the Position window. B. Drag the axis. If Snap is set to 5, the coordinate system moves in 5 unit increments.						
Specify a delta distance	Enter the Distance in the Position window. To translate along the same axis again, click a different manipulator feature, and then click the axis and enter the new delta value.						
Translate by typing new X, Y, Z coordinates	a. If the Position window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click one of the manipulator arcs or the sphere at the manipulator origin. c. Enter new X, Y, Z coordinate values.						
Rotate around an axis	Drag one of the small spheres.						
Rotate a specific number of degrees around an axis	a. If the Position window is not visible, right-click the manipulator and choose Show Position/Rotation window. b. Click one of the small spheres.						

To	Do this						
	c. Do one of the following: <table> <tr> <th>To</th><th>Do this</th></tr> <tr> <td>Drag using a snap increment</td><td> A. (Optional) Type a new Snap value in the Position window. B. Drag the sphere. If Snap is set to 15, the coordinate system rotates in 15 degree increments. </td></tr> <tr> <td>Specify a delta angle</td><td> Enter the Angle in the Position window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value. </td></tr> </table>	To	Do this	Drag using a snap increment	A. (Optional) Type a new Snap value in the Position window. B. Drag the sphere. If Snap is set to 15, the coordinate system rotates in 15 degree increments.	Specify a delta angle	Enter the Angle in the Position window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value.
To	Do this						
Drag using a snap increment	A. (Optional) Type a new Snap value in the Position window. B. Drag the sphere. If Snap is set to 15, the coordinate system rotates in 15 degree increments.						
Specify a delta angle	Enter the Angle in the Position window. To rotate along the same axis again, click a different manipulator feature, and then click the sphere and enter the new delta value.						
Align to another coordinate system	Do one of the following: <table> <tr> <th>To</th><th>Do this</th></tr> <tr> <td>Select a coordinate system in the Viewing window</td><td> a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window. </td></tr> <tr> <td>Select a coordinate system from a list</td><td> a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK. </td></tr> </table> The coordinate system moves and rotates to align to the coordinate system.	To	Do this	Select a coordinate system in the Viewing window	a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window.	Select a coordinate system from a list	a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK.
To	Do this						
Select a coordinate system in the Viewing window	a. Right-click the manipulator and choose Align to Coordinate System→Select in View. b. Click a coordinate system in the Viewing window.						
Select a coordinate system from a list	a. Right-click the manipulator and choose Align to Coordinate System→Select from List. b. In the Select Coordinate System dialog box, select a coordinate system and click OK.						
Snap to feature geometry	The coordinate system moves and rotates to align to the feature geometry.						

To	Do this	
	To	Do this
	To snap to the coordinate system of a Part	a. Right-click the manipulator and choose Snap to Feature Geometry→Part . b. Click the part.
	To snap to a point on a Face	a. Right-click the manipulator and choose Snap to Feature Geometry→Face . b. Click a point on the face.
	To snap to an endpoint of an edge	a. Right-click the manipulator and choose Snap to Feature Geometry→Edge . b. Click the edge, near the endpoint.
	To snap to any point on and edge	a. Right-click the manipulator and choose Snap to Feature Geometry→On Edge . b. Click a point on the edge.
	To snap to the midpoint of an edge	a. Right-click the manipulator and choose Snap to Feature Geometry→Edge Midpoint . b. Click the edge.
	To snap to any point on a part	a. Right-click the manipulator and choose Snap to Feature Geometry→Point . b. Click a point on the part.
Do one of the following:		
Rotate by aligning a manipulator axis to an edge or a plane to a surface	Right-click the manipulator and choose Align Axis .	
	To	Do this
	Align a manipulator axis to an edge on the model	a. Right-click the manipulator axis and choose Align Axis .

To	Do this	
	To	Do this
		b. Click the edge.
	Align a manipulator plane to a surface on the model	a. Right-click the small sphere on the manipulator plane and choose Align Plane . b. Click the surface.
The coordinate system rotates to align to the edge or surface.		

5. To turn off a manipulator, do any of the following:

- Right-click the manipulator and turn off **Manipulate**.
- Select the coordinate system, and then on the **3D Coordinate System** toolbar, click **Manipulate Selected Coordinate System** .

Edit a coordinate system

Edit Coordinate System

×

Name

Csys1

Align to Feature

Align Only:

Pick Feature

Align and Associate:

Pick Feature

Part Association:

Remove

Position

X:

0.000000

Y:

0.000000

Z:

0.000000

Orientation (degrees)

I:

-0.000000

J:

-0.000000

K:

-0.000000

OK

Cancel

Apply

1. Do one of the following:

To	Do this
Select a visible coordinate system	In the Viewing window, right-click a coordinate system and choose Edit .
Edit the selected coordinate system	- If it is not already selected, select the coordinate system. - From the 3D Coordinate System toolbar, click Edit Selected Coordinate System .
Select the coordinate system from the list	- On the 3D Coordinate System toolbar, click Manage Coordinate Systems . - In the Manage Coordinate Systems dialog box select the name of the coordinate system, and then click Edit.

A coordinate system manipulator appears on the coordinate system.

- (Optional) Use the **manipulator** to reposition the coordinate system.
- (Optional) In the **Edit Coordinate System** dialog box do any of the following:

To	Do this
Rename the coordinate system	In the Name section, type a new name.
Align the coordinate system with a part feature	In the Align to Feature section, next to Align Only, click Pick Feature, and then click a part feature in the Viewing window. The coordinate system moves to align with the part feature.
Align the coordinate system with a part feature and associate the coordinate system with the part	In the Align to Feature section, next to Align and Associate, click Pick Feature, and then click a feature on the part. The coordinate system moves to align with the feature, and the name of the associated part appears in the Part Association section. If you transform the part, the coordinate system will transform with it.
Specify coordinates of the origin	In the Position section, type X, Y, Z values.
Specify the orientation of the axes	In the Orientation section, type I, J, K values.

- Do one of the following:

To	Do this
Discard the changes before applying them	Click Cancel .
Apply the changes without closing the dialog box	Click Apply .
To accept the new coordinate system and close the dialog box	Click OK .

Choose the Working Coordinate System

The Working Coordinate System (Working CS) is used for **measuring** and can be used for **transforming** parts. You can choose any PMI or User Defined coordinate system to be the Working CS. If you do not designate a Working CS, the Global coordinate system is used for measurements.

Do one of the following:

To	Do this
Choose a visible coordinate system	In the Viewing window, right-click a coordinate system and choose Toggle Working CS .
Choose from the list of coordinate systems	1. On the 3D Coordinate System toolbar, click Manage Coordinate Systems . 2. In the Manage Coordinate Systems dialog box, select the name of a coordinate system, and then click Toggle Working CS.

Delete a coordinate system

Do any of the following:

To	Do this
Delete a visible coordinate system	In the Viewing window, right-click a coordinate system and choose Delete .
Delete the selected coordinate system	1. If it is not already selected, select the coordinate system. 2. On the 3D Coordinate System toolbar, click Delete Selected Coordinate System .
Delete a coordinate system from the list	1. On the 3D Coordinate System toolbar, click Manage Coordinate Systems . 2. In the Manage Coordinate Systems dialog box select the name of the coordinate system, and then click Delete.

Align a part or assembly to a coordinate system

1. Select the coordinate system.
2. Select the part or assembly.

Note:

You must select a single part or a single assembly. To align multiple parts that are not a single assembly, you must repeat the alignment process for each part.

3. Do one of the following:
 - In the Viewing window, right-click the coordinate system and choose **Align Selected Part to CS**.
 - On the **3D Coordinate System** toolbar, click **Align Part To Selected Coordinate System** .

Align your view to a coordinate system

1. Select the coordinate system.
2. Do one of the following:
 - In the Viewing window, right-click the coordinate system and choose **Align View to CS**.
 - On the **3D Coordinate System** toolbar, click **Align View To Selected Coordinate System** .

Set coordinate system appearance preferences

1. On the **3D Coordinate System** toolbar, click **Coordinate System Preferences** .
2. In the **Coordinate Systems Preferences** dialog box, in the **Coordinate System Type** section, select one of the following:
 - **User Defined**
 - **PMI**
 - **Selected**
 - **Working**
3. In the Display Attributes section, do any of the following:

To	Do this
Change the size of the trihedron	- In the Coordinate System Type section, select the type to change. - Drag the slider. The size (in pixels) appears next to the slider.
Change the color of the trihedron	- In the Coordinate System Type section, select the type to change. You cannot change the red, green, or blue colors of User Defined coordinate systems. - From the Color list, select a color.
Change the width of the trihedron axes	- In the Coordinate System Type section, select the type to change. - From the Axis width list, select a value.
Specify whether you can display name labels	- In the Coordinate System Type section, select User Defined or PMI. You cannot set this preference for Selected and Working coordinate systems. They inherit this preference from User Defined and PMI coordinate systems. - Select or clear the Enable labels check box. Note: If you select the Enable labels check box, you can right-click coordinate systems and choose to turn on or off Show Label.
Specify whether you can display axis planes	- In the Coordinate System Type section, select User Defined or PMI. You cannot set this preference for Selected and Working coordinate systems. They inherit this preference from User Defined and PMI coordinate systems. - Select or clear the Enable axis planes check box.

To	Do this
	Note: If you select the Enable axis planes check box, you can right-click coordinate systems and choose to turn on or off Show Planes.
Save the preference changes as the default preferences	Select the Set as default check box.
Specify how many coordinate systems can be displayed simultaneously	Type a value for Max. number of Coordinate Systems to display .

Select coordinate system features when selecting part features

You can select a coordinate system axis or axis endpoint for any task that involves selecting part features. Such tasks include **measuring parts**, **aligning cross sections**, and **rotating and translating parts**.

1. Start a task, like measuring parts, that involves selecting part features.
2. Do one of the following:

To select a	Do this
Coordinate system origin or an axis endpoint	On the 3D Selection toolbar , click Vertex  .
Coordinate system axis	On the 3D Selection toolbar , click Edge  .

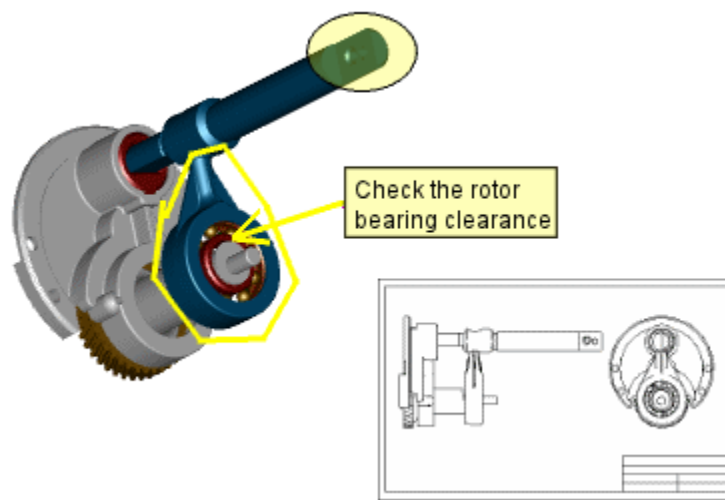
3. Click the coordinate system entity.

8. Marking up 3D models

Working with 3D markups

Overview of 3D markups

3D markups are 2D — or 3D when using the **In-Plane** feature — text or graphical elements that you draw in the Viewing window when working with a 3D model. Markups include different types of lines and geometric shapes, text, and inset images. End users typically use markups to add information to a 3D model.



The following apply to all 3D markups:

- Markups can either be **anchored** to a part or **floating** in the Viewing window.
- You can change the color of markups, the transparency of markups, and more, by specifying **markup preferences**.
- Markups are stored on **markup layers**. You can turn on and off markup layers or individual markups.
- When you **capture** or export an image of your Viewing window, the image includes all visible markups.
- When you save a session that includes 3D markup layers, the layer information is saved and associated with the work session. If you open a session file that includes a layer that has been modified since the session file was saved, a message will appear that indicates that the layer does not match the layer in the session file.

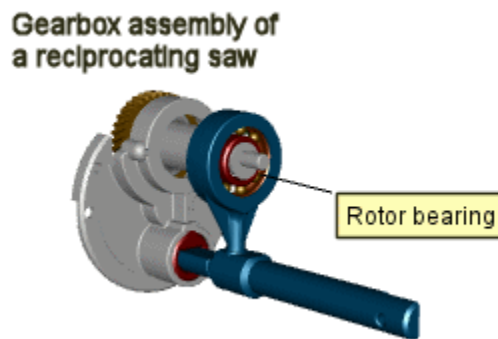
Overview of anchored and floating markups

All 3D markups are either anchored or floating.

- Anchored** Anchored markups are associated with specific parts and move to follow the parts as you navigate around the model.
- Anchored text and inset image markups are attached to parts with leaders.
 - Anchored markups of other types are attached directly to the parts.
 - By default, most anchored markups scale with the model; however, to scale anchored text markups that are not in **In-Plane** mode, you must set the **Zoom with view** preference.
 - By default, anchored markups are **displayed and hidden** with their attached parts.
- Floating** Floating markups are not associated with specific parts, and they remain where they were drawn in the Viewing window as you navigate around the model.

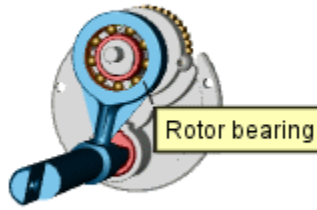
Example:

You add a floating text markup that says *Gearbox assembly of a reciprocating saw*, and an anchored text markup that says *Rotor bearing*.



When you rotate your view, the floating text markup remains where it is in the Viewing window (unless **In-Plane** mode is selected), and the anchored text markup moves with the part, adjusting its location and leader as necessary.

Gearbox assembly of
a reciprocating saw



Note:

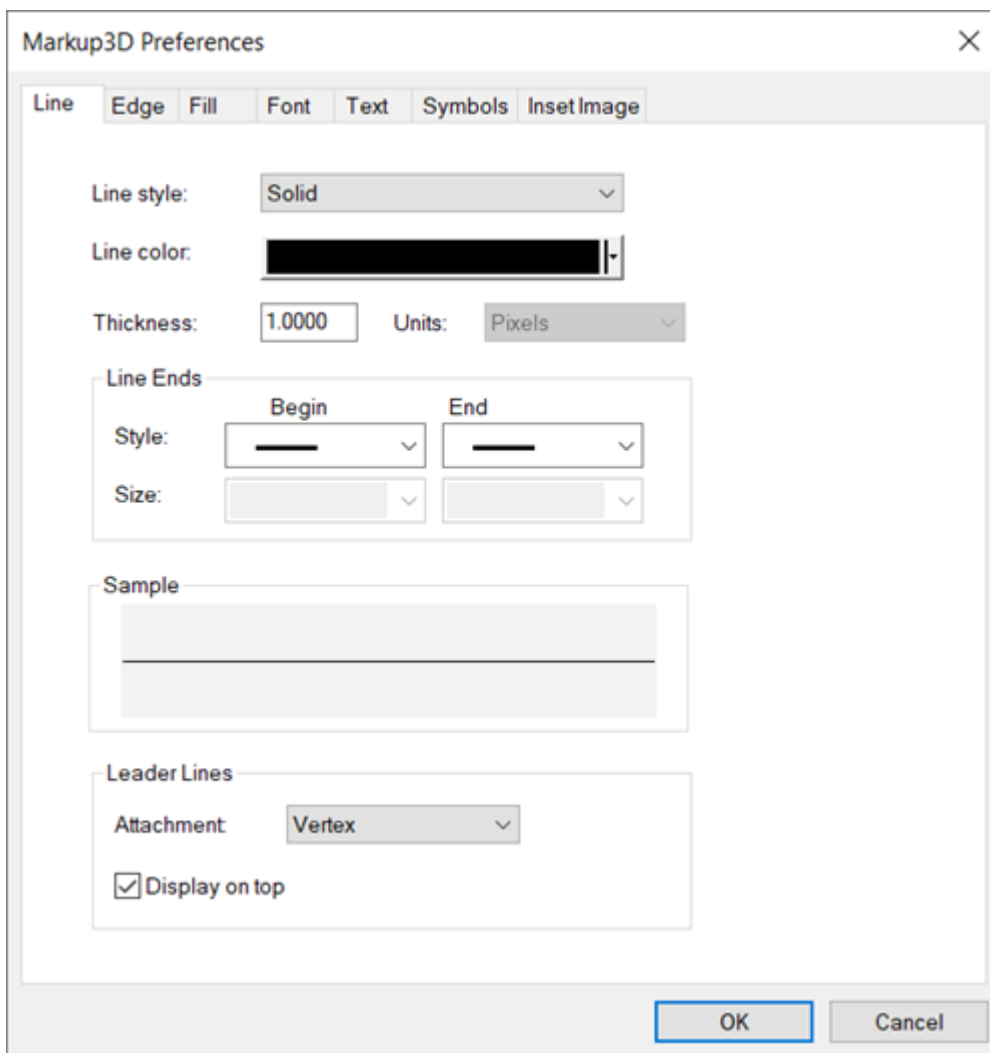
You can set a property or preference to specify the visibility behavior of anchored text upon loading a session or PLM XML file. You can specify whether to make the text visible when it is anchored to parts that have been removed from the model or moved to a different assembly in the same model. When anchored text is visible for missing parts, leader lines do not appear and the text appears in red.

Setting 3D markup preferences

Set 3D markup line preferences

Line preferences affect line markups and leaders to anchored text and inset image markups.

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Line** tab.
3. Choose any of the following options.



Use this option	To
Line style	Specify the line style.
Line color	Specify the line color.
Thickness	Specify the line thickness.
Units	Specify the units used for line thickness when page extents are active.
(Line Ends) Style	Specify whether arrowheads (or other symbols) are added to the ends of lines.
(Line Ends) Size	Specify the size of symbols added to the ends of lines.
Leader Lines	
Attachments	Specify how leaders must attach to anchored text or inset image markups. Choose from the following options:

Use this option	To
	• Edge Midpoint • Vertex • Center Note: If you choose a speech bubble as the bounding shape for your text markup, the leader attachment options have no effect. If you choose a circle as the bounding shape for your text markup, Center is the only leader attachment option available.
Display on Top	Specify whether leaders must always be drawn on top of the model in the 3D view. Choose this option to retain the leader position on top when the view is rotated.

Set 3D markup edge preferences

Edge preferences affect ellipses, rectangles, polygons, and text markup bounding shapes.

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Edge** tab.
3. Do any of the following:

Use this option	To
Edge style	Specify the edge style.
Edge color	Specify the edge color.
Thickness	Specify the edge thickness.
Units	Specify the units used for the edge thickness when page extents are active.

Set 3D markup fill preferences

Fill preferences affect ellipse, rectangle, polygon, and text markup bounding shapes created with **Fill Mode**  enabled.

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Fill** tab.
3. Do any of the following:

Use this option	To
Fill style	Specify a line pattern used to fill shapes when fill mode is enabled.
Fill color	Specify the color used to fill shapes when fill mode is enabled.
Transparency	Make the fill more or less transparent. Drag the slider to the left to make the fill more transparent or to the right to make the fill more opaque.

Set 3D markup font preferences

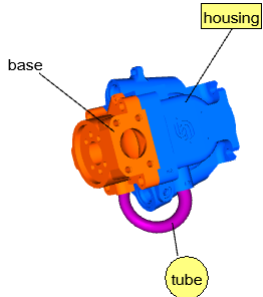
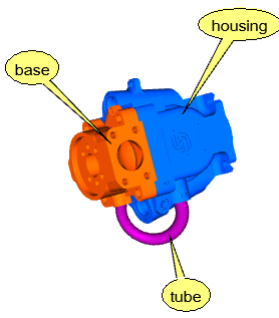
Font preferences affect floating text, anchored text, callouts, and pre-defined text markups.

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Font** tab.
3. Do any of the following:

Use this option	To
Font	Select the name of the font you want to use.
Font Style	Specify Regular , Bold , Bold Italic , or Italic . (Some fonts have only Regular style.)
Size	Select the point size of the font you want to use.
Underline	Underline text in text markups.
Strikeout	Draw a line through text in text markups.
Color	Select the color of the font you want to use.

Set 3D markup text preferences

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Text** tab.
3. Do any of the following:

In this section	Use this option	To
Text Alignment	Horizontal	Specify Left , Center , or Right .
	Vertical	Specify Top , Center , or Bottom .
Bounding Shape	Bounding Shape	Specify None, Rectangle, Circle, or Speech Bubble. Note: When using the Rectangle, Circle, or None bounding shapes, the leader anchoring a text markup to a part is a simple line. When using the Speech Bubble bounding shape, the leader is the tail of the bubble.  <i>None, Rectangle or Circle</i>  <i>Speech bubbles</i>

In this section	Use this option	To
Bounding Shape Size (This section is available when page extents are active.)	Size to fit text	Select or clear the check box to control whether the bounding shape resizes to fit text when page extents are active.
	Width	Specify the width of the shape if Size to fit text is not active and page extents are active.
	Height	Specify the height of the shape if Size to fit text is not active page extents are active.
	Units	Specify the units for width and height when page extents are active.
Anchored Text	Zoom with view	Specify whether to scale anchored text markups when the model is scaled, for example, when using zoom options.
	Visible when anchored parts are not found	Specify whether to make anchored text visible (upon loading a session or PLM XML file) when it is anchored to parts that have been removed from the model or moved to a different assembly in the same model. When anchored text is visible for missing parts, leader lines do not appear and the text appears in red.
Automatic Creation	Create on	Specify which parts to add text to when using Auto Create to add anchored text. Options are Selected Parts and Visible Parts .
	Prompt at creation time	Specify whether to display the Anchored Text Automatic Creation dialog box when you use the Auto Create command. In the dialog box, you specify to generate anchored text markups on all Selected Parts or Visible Parts .
Pre-defined Text	Include labels	Specify whether labels appear for Location, Part Information, and Metadata Attributes text.
	Pre-defined Text	Set the predefined text preferences.
Callouts	Callouts	Set 3D markup callout preferences.

- Click **OK**.

Set 3D markup pre-defined text preferences

Define the text to add when **Use Pre-defined Text Mode** is active.

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, on the **Text** tab, in the **Pre-defined Text** section, click **Edit**.
3. In the **Edit Text** dialog box, do any of the following:

To	Do this
Type text	Type in the right-hand section of the dialog box.
Start a new line	Click to position the text cursor and press Enter.
Delete text	Drag to select the text and press Delete.
Move text	Drag to select the text, and then cut and paste.
Add a callout	Select <Callout> and click Add Key .
Add part information (to anchored text only)	a. Click the + symbol to expand Part Information. b. Select any of the following: • <Part name> • <Part Heritage> • <Parent Assembly> c. Click Add Key to add a placeholder for the attribute. d. Click Add Value to add a placeholder for the value.
Add metadata attributes (to anchored text only)	The list of metadata attributes varies depending on the information associated with model. a. Click the + symbol to expand Meta Data Attributes. b. Select an attribute. c. Click Add Key to add a placeholder for the attribute.

To	Do this
	d. Click Add Value to add a placeholder for the value.
Specify which metadata attributes to list	a. Click Attributes List. b. In the Attribute Selection dialog box, from the Source list, choose one of the following: - To select attributes stored in the JT, choose JT. (For parts loaded from Teamcenter) To select Teamcenter attributes, choose Teamcenter Structure Lines. Note: Teamcenter Integration Attributes Preferences control which Teamcenter attributes appear when you choose Teamcenter Structure Lines. c. Click Execute. d. Click OK.

- If you are using **3D collaborative review and markup**, and you have selected an author layer in the layer list, you have the option to add **Author Profile Information**. The profile information varies depending on the information you have defined.
- Click **OK**.

The text appears in the **Pre-defined Text** section of the **Markup 3D Preferences** dialog box.

- Select or clear the **Include Labels** check box to specify whether labels appear for Location, Part Information, and Metadata Attributes text.

Note:

Pre-defined text appears as the default text in **Text Markup** dialog box and is used when **Use Pre-defined Text Mode** is active.

Set 3D markup callout preferences

- From the **3D Markup** toolbar, select **Preferences** .
- In the **Markup 3D Preferences** dialog box, on the **Text** tab, click **Callouts**.

3. In the **Callouts** dialog box, do any of the following:

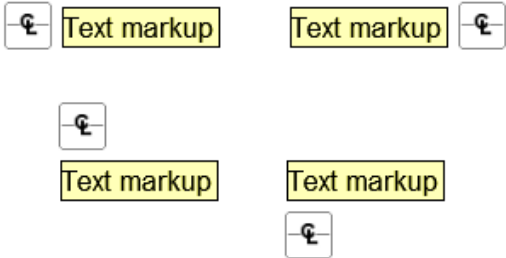
Use this option	To
Default Callout Format	Specify whether callouts are Numeric or Alphabetic .
Starting Value	Specify the starting number or letter.
Resequence Collision Action	Specify what to do if you find duplicate callout values when using the Resequence Callouts command. Choose one of the following: • Ignore to allow two callouts to have the same value. • Reject to cancel the resequencing operation when a duplicate value is found. • Show Warning to display the Resequence Collisions Warning dialog box when a duplicate value is found. Note: If you select Reject or Show Warning, collision detection is performed on future resequencing operations. It does not detect duplication of callouts that already exist and which are not part of a resequencing.

4. Click **OK**.

Set 3D markup symbol preferences

- From the **3D Markup** toolbar, select **Preferences** .
- In the **Markup 3D Preferences** dialog box, click the **Symbols** tab.
- Do any of the following:

In this section	Use this option	To
Symbol Placement	Symbol Location	Specify where to place symbols added to 3D text markups. Options are Top , Bottom , Left , and Right .

In this section	Use this option	To
		Example: For the first two text markups below, Symbol Location is Left and Right respectively. For the bottom two text markups, Symbol Location is Top and Bottom, respectively. 
Symbol Stacking	Stack Symbols	Specify whether to create a single row of symbols, or multiple <i>stacked</i> rows based on the Symbols Per Row value.
	Symbols Per Row	Specify maximum number of symbols in a row when Stack Symbols is active. Example: The first six symbols below are in a in single row. The second six symbols are stacked in three rows, with two symbols in each row. 
Symbol Size	Width	Specify the symbol width.
	Height	Specify the symbol height.
	Spacing	Specify the spacing between symbols.
	Units	Specify the units for width, height, and spacing.

In this section	Use this option	To
		Note: To have units other than pixels, you must enable page extents.
Symbol Library Location	Path	Specify the path to the symbol library folder. Click Browse to change.

4. Click **OK**.

Set 3D markup inset image preferences

1. From the **3D Markup** toolbar, select **Preferences** .
2. In the **Markup 3D Preferences** dialog box, click the **Inset Image** tab.
3. Do any one of the following.

Use this option	To implement this insertion behavior
Size manually (default option)	Size the image manually at the time of insertion. Note: If the property Size manually is selected, and an inset image is anchored to a part, the image scales with the model when you use zoom options.
Size automatically	
Original size	Insert an image at its fixed, original size.
Width and Height	Insert an image with the specified width and height. Note: The Units value is Pixels unless you enable page extents. If you enable page extents,

Use this option	To implement this insertion behavior
	then ensure to select an appropriate unit type.

Note:

Altering this setting in the preferences, affects inset image markups added to the view *after* you make this modification.

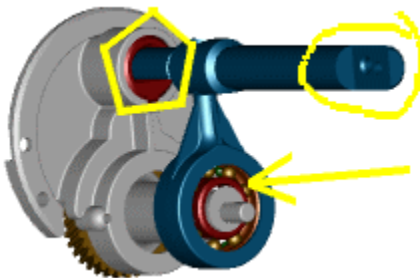
To modify an existing inset image, you must select the appropriate inset image, and **edit its properties**.

Drawing 3D markups

Add straight line, freehand line, and polyline markups

Example:

For the model below, an arrow points to the rotor bearing assembly, a freehand line markup indicates part of the spindle, and a polyline markup identifies the keyway.



1. (If it is not displayed already) Display the **3D Markup** toolbar.
2. (If the toolbar buttons are not active) Click **Enable Markup** .
3. (Optional) **Create** a new layer, or select an existing layer to add the markup to.
4. (Optional) Change the line type, color, thickness, and ending style by setting **markup preferences**.
5. (Optional) To place the line markup in the same plane as the model, which allows for more precise tracking of the markup, on the **3D Markup** toolbar, click **In-Plane** .

Note:

In-Plane mode remains activated for all markups you add after this step, until you deactivate it.

6. (Optional) To associate the line markup to a specific model part, on the **3D Markup** toolbar, click **Anchor** mode .

Note:

Anchor mode remains activated for all markups you add after this step, until you deactivate it.

7. On the **3D Markup** toolbar, click the line tool for the markup you want to add.

The cursor changes to indicate the type of markup.

Note:

By default, each markup you draw is placed on the currently active layer. If no layer exists when a markup is created, a new layer is created automatically and the markup placed into it.

8. In the Viewing window, if **In-Plane** or **Anchor** modes are selected, select a point on a part.
9. Draw the markup using the appropriate steps below.

To place this markup	Do this
Freehand Line 	Click and drag the cursor in the Viewing window.
Line 	Click and drag the cursor in the Viewing window.
Polyline 	- Click in the Viewing window to place one endpoint. - Move the cursor and click to place the next vertex in the polyline. - Repeat the previous step as many times as needed. - Double-click to place the last endpoint of the polyline.

Tip:

To draw an exact horizontal or vertical lines, hold the Shift key as you drag the cursor.

The markup appears in the Viewing window, as well as in the list of markups under the selected markup layer. A * appears before the markup layer name to indicate that changes have been made to the markup layer since it was last saved.

- (Optional) To rename the markup, in the **Markup Layers** list, right-click the markup, click **Rename**, and then type the name.

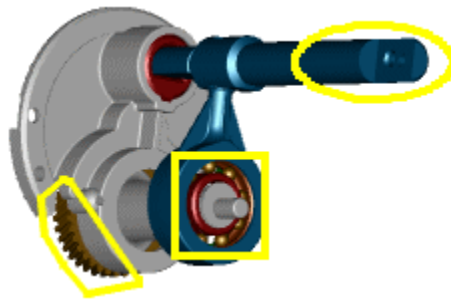
The markup appears on the selected layer in the **Markup Layers** list and in the Viewing window.

Add rectangle, ellipse, and polygon markups

Use **markups** to add rectangles, ellipses, and polygons to your 3D models in the Viewing window.

Example:

For the model below, a rectangle markup indicates the rotor bearing assembly, an ellipse markup encircles part of the spindle, and a polygon markup identifies visible portion of the gear.



- (If it is not displayed already) Display the **3D Markup** toolbar.
- (If the toolbar buttons are not active) Click **Enable Markup** .
- (Optional) To draw a filled markup, on the **3D Markup** toolbar, click **Fill Mode** .

Note:

Fill mode remains activated for all markups you add after this step, until you deactivate it.

- (Optional) **Create** a new layer, or select an existing layer to add the markup to.
- (Optional) Change the appearance parameters for the markup by setting **markup preferences**.
- (Optional) To place the geometric markup in the same plane as the model, which allows for more precise tracking of the markup, on the **3D Markup** toolbar, click **In-Plane** .

Note:

In-Plane mode remains activated for all markups you add after this step, until you deactivate it.

7. (Optional) To associate the geometric markup to a specific model part, on the **3D Markup** toolbar, click **Anchor** mode .

Note:

Anchor mode remains activated for all markups you add after this step, until you deactivate it.

8. On the **3D Markup** toolbar, click the tool for the markup you want to add.

The cursor changes to indicate the type of markup.

Note:

By default, each markup you draw is placed on the currently active layer. If no layer exists when a markup is created, a new layer is created automatically and the markup placed into it.

9. In the Viewing window, if **In-Plane** or **Anchor** modes are selected, select a point on a part.
10. Draw the markup using the appropriate steps below.

To place this markup	Do this
Rectangle 	- Click in the Viewing window to place one corner. - Drag the cursor, and then release to place the opposite corner.
Polygon 	- Click in the Viewing window to place one vertex. - Move the cursor and click to place the next vertex in the polygon. - Repeat the previous step as many times as needed. - Double-click to place the last vertex in the polygon.
Ellipse 	- Click in the Viewing window to place one outer edge. - Drag the cursor to define the diameter of the ellipse, and then release to place the opposite edge.

Tip:

To draw an exact square, horizontal or vertical polygon segment, or circle; hold the Shift key as you drag the cursor.

The markup appears in the Viewing window, as well as in the list of markups under the selected markup layer. A * appears before the markup layer name to indicate that changes have been made to the markup layer since it was last saved.

11. (Optional) To rename the markup, in the **Markup Layers** list, right-click the markup, click **Rename**, and then type the name.

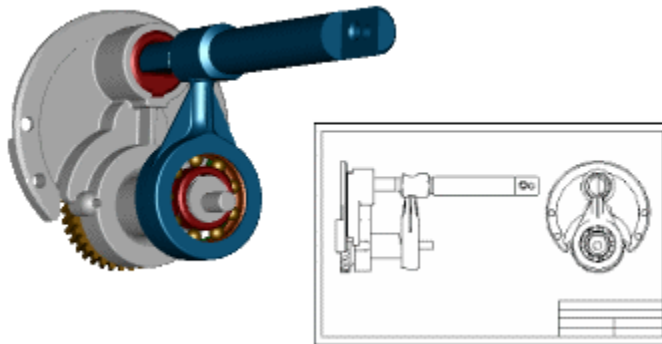
The markup appears on the selected layer in the **Markup Layers** list and in the Viewing window.

Add inset image markups

You can use 3D inset image markups to include 2D images in the 3D Viewing window.

Example:

For the model below, a floating inset image of the blueprint for the part appears next to the 3D model of the part.



1. (Optional) **Create** a new layer, or select an existing layer to add the markup to.
2. On the **3D Markup** toolbar, do the following:
 - Turn on **3D Markup** .
 - (Optional) Click **Preferences**  and change your preferences.

Note:

If the property **Size manually** is active for an anchored inset image, the image scales with the model when you use zoom options.

- (Optional) To associate the inset image markup to a specific model part, make sure **Anchor** mode  is activated.

Note:

Anchor mode remains activated for all markups you add after this step, until you deactivate it.

- (Optional) To place the inset image markup in the same plane as the model, which allows for more precise tracking of the markup, on the **3D Markup** toolbar, click **In-Plane** .

Note:

In-Plane mode remains activated for all markups you add after this step, until you deactivate it.

3. Click **Inset Image** .

Note:

By default, each markup you draw is placed on the currently active layer. If no layer exists when a markup is created, a new layer is created automatically and the markup placed into it.

4. In the **Insert Inset Image** dialog box, choose an image, and then click **Open**.
5. In the Viewing window, do one of the following:
 - If **In-Plane** or **Anchor** mode is selected, select a point on a part.
 - If neither **In-Plane** or **Anchor** mode is selected, select any point in the Viewing window.
6. If the **Inset Image** preference **Size manually** is active, you must click a second time and drag to size the image. Otherwise, based on the selection in the **Size automatically** area, either one of the following take place:
 - The image is inserted with its **Original size**.
 - The image is inserted with the specified **Width** and **Height**.

The **Units** value is **Pixels** unless you enable page extents. If you enable page extents, then ensure to select an appropriate unit type.

7. (Optional, anchored inset image) Drag the inset image away from the part to see the leader attached to the inset image and the part.
8. (Optional, anchored inset image) To add another leader to the markup, do the following:
 - a. Right-click the inset image markup and choose **Add Leader Line**. The add leader mode is enabled.
 - b. Click the next part, or another point on the same part, to which you want to add a leader.

A leader is added to the markup.

- c. Disable the add leader mode by right-clicking the inset image markup again and choosing **Add Leader Line**. The add leader mode is now toggled off.

Add text markups

1. On the **3D Markup** toolbar, do the following:
 - Turn on **3D Markup** .
 - (Optional) Click **Preferences**  and change your preferences.
 - (Optional) To associate text markup to a specific model part, turn on **Anchor** mode .

Note:

Anchor mode remains activated for all markups you add after this step, until you deactivate it.

By default, most anchored markups scale with the model; however, to scale anchored text markups that are not in **In-Plane** mode, you must set the **Zoom with view** preference.

- (Optional) To place the text markup in the same plane as the model, which allows for more precise tracking of the markup, on the **3D Markup** toolbar, click **In-Plane** .

Note:

In-Plane mode remains activated for all markups you add after this step, until you deactivate it.

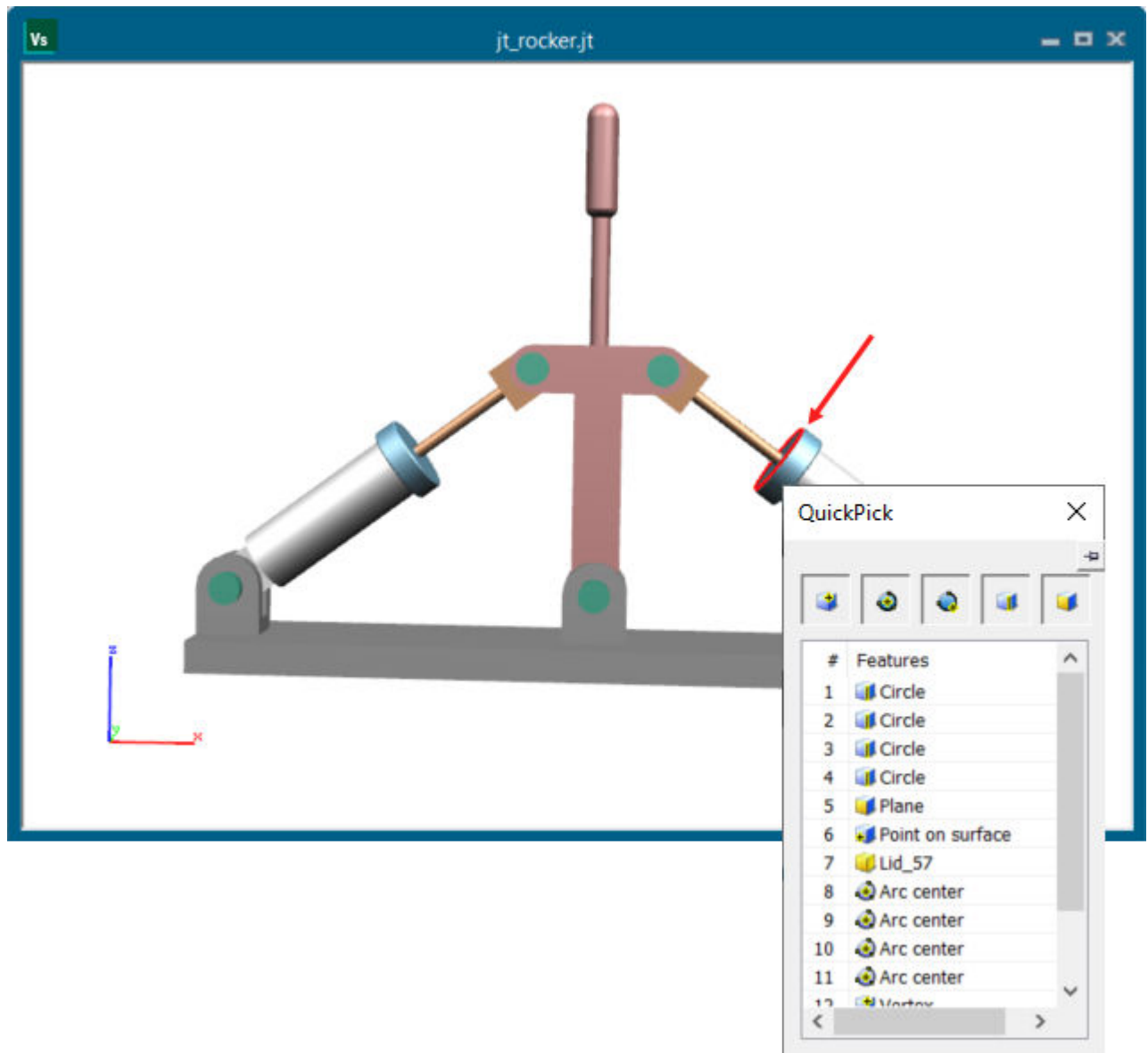
2. (Optional) **Create** a new layer, or select an existing layer to add the markup to.

3. Turn on **Text**  mode.

Note:

By default, each markup you draw is placed on the currently active layer. If no layer exists when a markup is created, a new layer is created automatically and the markup placed into it.

4. In the Viewing window, do one of the following:
 - If **In-Plane** or **Anchor** mode is selected, select a point on a part.
 - If neither **In-Plane** or **Anchor** mode is selected, select any point in the Viewing window.
5. (Optional) To pick a specific point on a part feature, such as the part edge, do the following.
 - Hover over the part feature until the feature turns red.
 - When three dots appear next to the pointer , click to display the **QuickPick** dialog box.



- Select the desired anchor location.

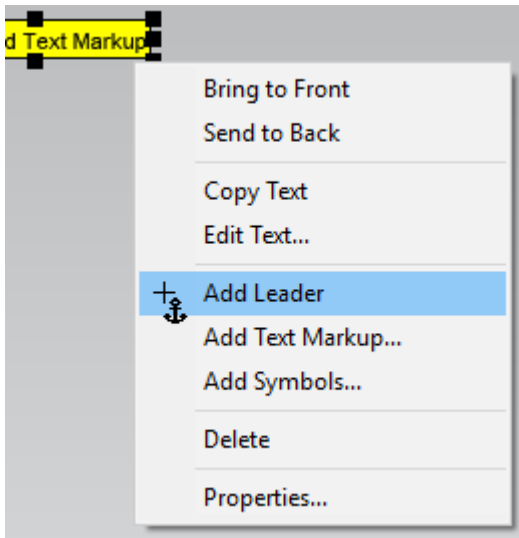
Note:

Turn off individual section filters to constrain the options in the **QuickPick** dialog box.

6. In the **Markup Text** dialog box, add or edit the markup text using **Basic** or **Advanced** mode.
7. Click **OK**.

The text markup appears in the Viewing window and on the list of markups under the active markup layer.

8. (Optional, anchored text) Drag the text away from the part to see the leader attached to the text and the part.
9. (Optional, anchored text) To add another leader to the markup, do the following:
 - a. Right-click the text markup and choose **Add Leader**. The add leader mode is enabled.



- b. Click the next part, or another point on the same part, to which you want to add a leader.

A leader is added to the markup.

- c. Disable the add leader mode by right-clicking the text markup again and choosing **Add Leader**. The add leader mode is now toggled off.

Add in-plane markups

Use the **In-Plane**  feature to allow for precise tracking with the markup's original position to the model.

1. On the **3D Markup** toolbar, do the following:

- Turn on **3D Markup** .
- Choose **In-Plane** .

2. Do one of the following:

To add a floating in-plane markup	To add an anchored in-plane markup
a. On the 3D Markup toolbar, choose the tool for the markup you want to add, such as Ellipse, Text, or Polygon. The cursor changes to indicate the type of markup b. Click on a model part to determine the plane to use for this markup.	a. On the 3D Markup toolbar, choose Anchor . b. Choose the tool for the markup you want to add, such as Ellipse, Text, or Polygon. The cursor changes to indicate the type of markup c. In the Viewing window, click the point on the part where you want to anchor the markup.

- Click in the Viewing window and draw the markup in the window background.

Drawing actions differ between the markup tools for **lines**, **geometric shapes**, **inset images**, or **text markup**.

Re-anchor anchored text or inset image leader lines

Use the following procedure to re-anchor an existing anchored text or inset image's leader line.

- In the viewing window, right-click the leader line and select **Re-Anchor Leader**.
- Click a new point on the model to attach the leader.

Add BOM line properties to text markups in embedded viewers

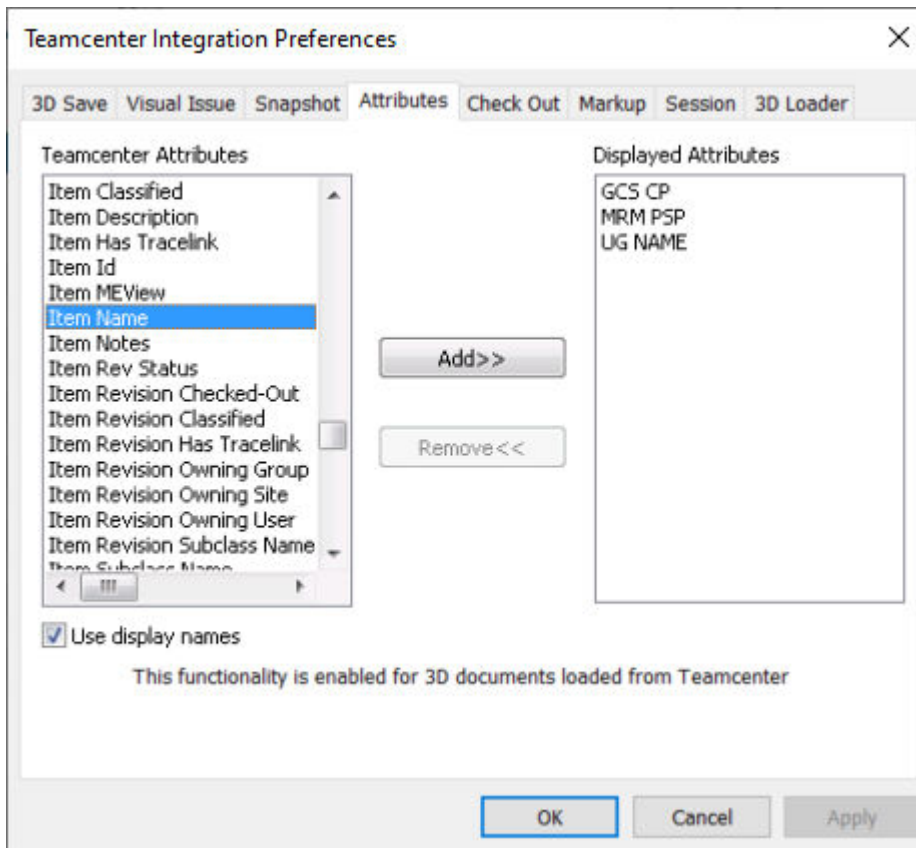
In the Lifecycle Viewer and standalone Lifecycle Visualization you can later load BOM line properties from Teamcenter and use the properties to populate anchored text markups. The markups act as captions or callouts displaying specific metadata from Teamcenter. Markups can be saved in visualization session files.

Before you can add BOM line properties to anchored text markups in embedded viewers, you must specify which properties should be available. You do that in the Lifecycle Viewer or standalone Lifecycle Visualization. The BOM line properties you specify are stored as the **Interop_Vis_Attributes** user preference.

Specify which BOM line properties can be added to anchored text markups

- Send an Item Revision to Lifecycle Viewer (or open it in Lifecycle Visualization).
- In the viewer, choose **File**→**Preferences**→**Teamcenter Integration**.

3. In the **Preferences** dialog box, click the **Attributes** tab.
4. From the **Teamcenter Attributes** list, select an attribute (a BOM line property) and click **Add**.



Note:

Administrators can use the **interop_vis_attributes_admin_filter** attribute to control which BOM line properties appear in the **Teamcenter Attributes** list.

5. Repeat the previous step for each attribute you want to be available for text markups.
6. Click **OK**.
7. Close the viewer.
8. Do the following to reload your Teamcenter preferences:
 - a. Choose **Edit→Options→Filters**.
 - b. In the **Options** dialog box, click **Filters** (bottom).
 - c. Click **Reloads the Site, Group, Role, and User preferences on the server**  (top).

- d. (Optional) To confirm that the preference is set correctly, type **interop** in the **Search by preference name** box, and select the **Interop_Vis_Attributes** User preference.

The BOM line attributes you selected in the Lifecycle Viewer should appear in the **Value** area on the right.

Add BOM line properties to 3D anchored text markups

You can add BOM line properties to any 3D anchored text markups. There are several ways to create 3D anchored text markups:

To	Do this
Manually add text markup to a part.	- On the 3D Markup toolbar, click 3D Markup , and then click Text  and Anchored , and (optionally) In-Plane . - Click a part in the viewing window. - If the Markup Text dialog box opens in Basic mode, click Advanced to switch to Advanced mode. - In the Edit Markup Text section, expand Meta Data Attributes. - Select an attribute and click Add <Key>. (Optional) Add more attributes and other text. (Optional) Select the Include labels check box. - Click OK. The text markup appears next to the part in the Viewing window.
Add predefined text to each part you click.	- On the 3D Markup toolbar, click 3D Markup  and Preferences . - In the Markup3D Preferences dialog box, click the Text tab. - In the Pre-defined Text section, click Edit. - In the Edit Markup Text section of the Edit Text dialog box, expand Meta Data Attributes. - Select an attribute and click Add <Key>.

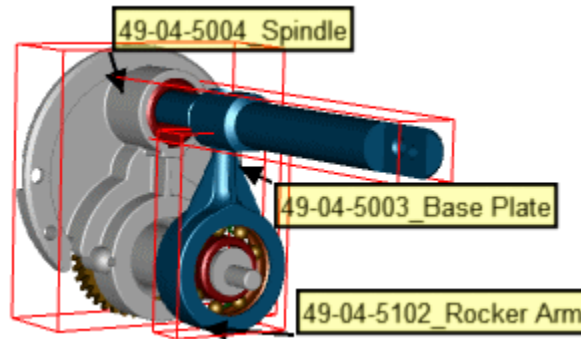
To	Do this
	(Optional) Add more attributes and other text. - Click OK. (Optional) On the Text tab of the Markup3D Preferences dialog box, select the Include labels check box. - Click OK. - On the 3D Markup toolbar, click Text , Anchor Mode , and Use Pre-defined Text Mode , and (optionally) In-Plane . - In the Viewing window, click each part to mark up. Markups are added as you click the parts.
Add predefined text to all selected parts with one click.	- On the 3D Markup toolbar, click 3D Markup  and Preferences . - In the Markup3D Preferences dialog box, click the Text tab. - In the Pre-defined Text section, click Edit. - In the Edit Markup Text section of the Edit Text dialog box, expand Meta Data Attributes. - Select an attribute and click Add <Key>. (Optional) Add more attributes and other text. - Click OK. (Optional) On the Text tab of the Markup3D Preferences dialog box, select the Include labels check box. - Click OK. - In the Viewing window, select the parts to markup. - On the 3D Markup toolbar, click Auto Create.  Markups are added to the selected parts.

Automatically add anchored text to multiple parts

Automatically created anchored text is useful when you want to add the same text markup to a number of parts at once.

Example:

For the model below, you select several parts, and then use **Auto Create** to add *<Part Name>* to all the parts at once.



1. On the **3D Markup** toolbar, turn on **3D Markup** .
2. (Optional) To change how you specify which parts the auto-created text is added to, do the following:
 - a. On the **3D Markup** toolbar, click **Preferences** .
 - b. In the **Markup 3D Preferences** dialog box, click the **Text** tab.
 - c. In the **Automatic Creation** section, do one of the following:
 - To have the **Automatic Creation** dialog box appear when using the **Auto Create** command, select the **Prompt at creation time** check box.
 - To set the default for which parts the text markups are automatically added to, select **Selected Parts** or **Visible Parts** from the **Create on** list.
 - d. Click **OK**.
3. Do one of the following:
 - (If adding text markups to all selected parts) Select the parts to add text to.
 - (If adding text markups to all visible parts) Turn on only the parts to add text to.

4. On the **3D Markup** toolbar, click **Auto Create** .
5. (If the **Anchored Text Automatic Creation** dialog box appears) Select **Selected Parts** or **Visible Parts** from the **Generate anchored text markups on** list, and click **OK**.
6. In the **Markup Text** dialog box, do any of the following:

To	Do this
Type text	Type in the right-hand section of the dialog box.
Start a new line	Click to position the text cursor and press Enter.
Delete text	Drag to select the text and press Delete.
Move text	Drag to select the text, and then cut and paste.
Add a callout	Select <Callout> and click Add Key .
Add part information (to anchored text only)	a. Click the + symbol to expand Part Information. b. Select any of the following: • <Part name> • <Part Heritage> • <Parent Assembly> c. Click Add Key to add a placeholder for the attribute. d. Click Add Value to add a placeholder for the value.
Add location (to anchored text only)	Include x, y, or z coordinate location information corresponding to the coordinate values of the anchor point for the text markup. a. Click the + symbol to expand Locations. b. Select a coordinate. c. Click Add Key to add a coordinate for the text markup. d. (Optional) Click Location... to customize the location information. If Include labels is selected on the Markup text dialog, the 3D anchored text shows the appropriate positive or negative value label for the x, y, and z coordinates based on whether

To	Do this
	a label is defined. You may also customize the precision and show the absolute value of the coordinates.
Add metadata attributes (to anchored text only)	The list of metadata attributes varies depending on the information associated with model. - Click the + symbol to expand Meta Data Attributes. - Select an attribute. - Click Add Key to add a placeholder for the attribute. - Click Add Value to add a placeholder for the value.
Specify which metadata attributes to list	- Click Attributes List. - In the Attribute Selection dialog box, from the Source list, choose one of the following: - To select attributes stored in the JT, choose JT. (For parts loaded from Teamcenter) To select Teamcenter attributes, choose Teamcenter Structure Lines. Note: Teamcenter Integration Attributes Preferences control which Teamcenter attributes appear when you choose Teamcenter Structure . - Click Execute. - Click OK.

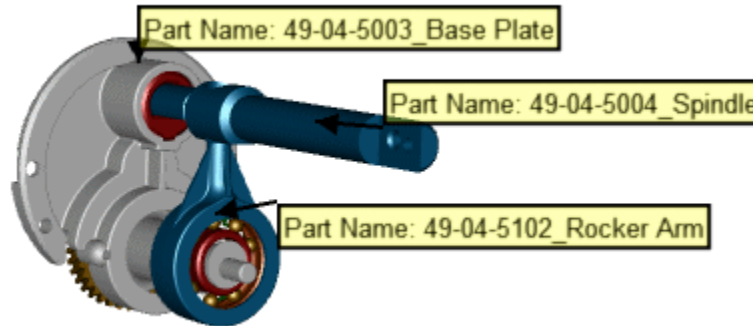
- If you are using **3D collaborative review and markup**, and you have selected an author layer in the layer list, you have the option to add **Author Profile Information**. The profile information varies depending on the information you have defined.
- Select or clear the **Include Labels** check box to specify whether labels appear for Part Information and Metadata Attributes text.
- Click **OK**.

Add predefined text markups

Predefined text is the default text used for 3D text markups. Your predefined text is used (unchanged) when you add a markup with **Use Pre-defined Text Mode** activated.

Example:

For the model below, if you set predefined text to be the attribute key <Part Name> and select **Use Pre-defined Text Mode**, you can click each part in turn to add the anchored text.



1. On the **3D Markup** toolbar, click **Enable Markup** .
2. (If you have not set predefined text) **Specify the predefined text**.
3. On the **3D Markup** toolbar, click **Use Pre-defined Text Mode** .

Note:

Predefined Text mode remains activated for all markups you add after this step, until you deactivate it.

4. Do any of the following:
 - Add **text** markups.
 - Use **Auto Create** to add **multiple** anchored text markups.

The text markups contain the predefined text.

Using part information, metadata, and anchor point location in text markups

All parts have part information attributes (for example, Part Name). They may also have metadata attributes (for example, Material). You can add part information, metadata attribute values, and anchor point location to text markups. Attributes consist of (*Key, Value*) pairs. Two examples are (*Part Name*,

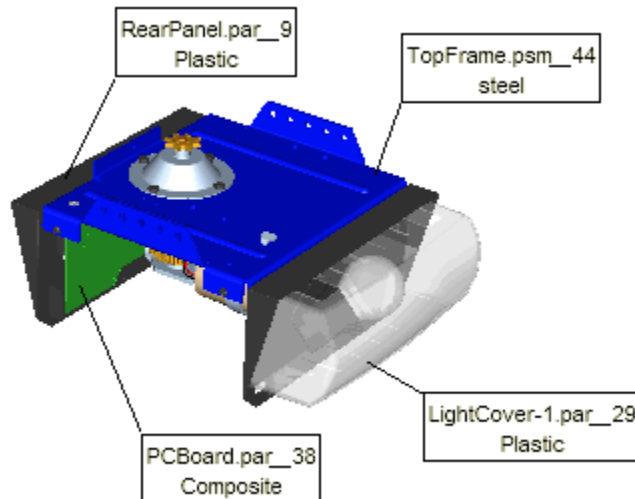
TopFrame) and (*Material, steel*). Metadata attributes vary depending on the information associated with your models. Part Information includes the following:

- **Part Name** is the name of the part as it appears in the assembly tree.
- **Parent Assembly** is the name of the subassembly of which the part is a member.
- **Part Heritage** is the part name, the name of its parent assembly, and the names of the assembly's parents up to the model name.

By using attribute *Keys* in markup text, you can create a markup *recipe* and use it with multiple parts.

Example:

If you set your pre-defined text to be <Part Name> and <Material>, and then add anchored text markups to several parts, each markup shows the part name and material values.

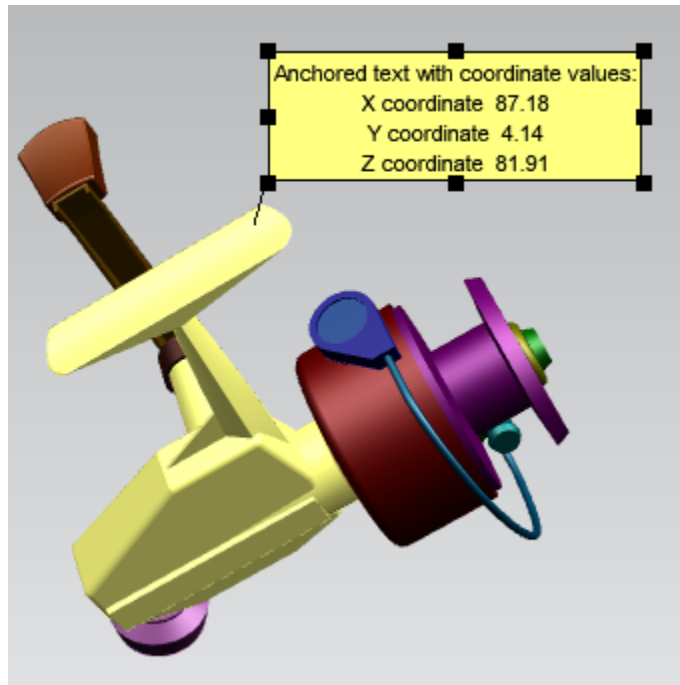


Note:

Attribute *Key* values are applied at the time the markups are created. Subsequent changes to metadata values on associated parts do not affect existing markups.

Use the x, y, and z coordinate location information to include the coordinate values of the anchor point for the 3D anchored text markup. You may also customize the location information or show the absolute value of the coordinates.

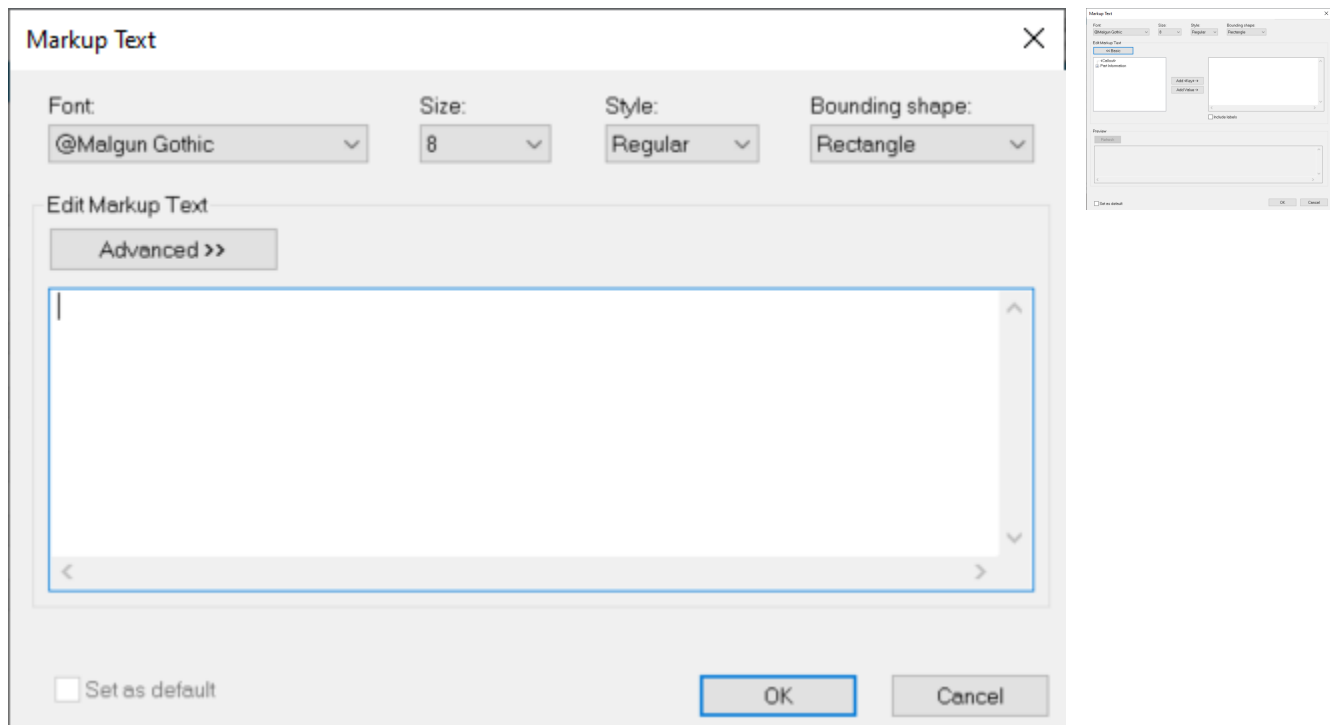
If **Include labels** is selected on the **Markup Text** dialog, the 3D anchored text shows the appropriate positive or negative value label for the x, y, and z coordinates based on whether a label is defined. You may also customize the precision and show the absolute value of the coordinates.



Defining markup text

Overview of defining markup text

The process of defining markup text is the same whether you are creating new markups or editing existing markups. You add or edit markup text in the **Markup Text** and **Edit Text** dialog boxes. In both dialog boxes, you can switch between **Basic** and **Advanced** modes. You can create and edit text markups in either mode, but Basic mode has a simple user interface and is recommended for most text markups. Switching to Advanced mode is recommended for adding callouts, part names, and metadata attributes. You can switch between Basic and Advanced at any time.



Add or edit markup text in Basic mode

1. Do one of the following:
 - Start to add a new text markup (with **Use Pre-defined Text Mode** deactivated).
 - Edit an existing text markup.

The **Text Markup** or **Edit Text** dialog box appears.

2. (If Advanced mode is active) Click **Basic**.
3. Do any of the following:

To	Do this
Type text	Type in the Edit Markup Text section.
Start a new line	Click to position the cursor and press Enter.
Delete text	Drag to select the text and press Delete.
Move text	Drag to select the text, and then cut and paste.

To	Do this
Copy text	Drag to select the text, and then copy and paste.
Change the font, size, style, or bounding shape	Select from the Font , Size , Style , or Bounding shape lists

- (Optional) To save the new text, attributes, and mode as the default, select the **Set as default** check box.
- Click **OK**.

Note:

Switching to Advanced mode is recommended for adding callouts, part information, and metadata attributes, but you can also add them in Basic mode by typing the correct key. For example, type `<Callout>` to add a callout.

Tip:

- To make Basic mode the default (after changing it to Advanced), create a new text markup using Basic mode and select the **Set as default** check box before you click **OK**.
- To clear the default text, edit the **pre-defined text** on the **Text** tab of the **Markup 3D Preferences** dialog box.

Add or edit markup text in Advanced mode

Advanced mode is recommended for adding callouts, part information, anchor point coordinate information, or metadata attributes to text markups.

- Do one of the following:
 - Start to add a new text markup (with **Use Pre-defined Text Mode** deactivated).
 - Edit an existing text markup.

The **Text Markup** or **Edit Text** dialog box appears.

- (If Basic mode is active) Click **Advanced**.
- Do any of the following:

To	Do this
Type text	Type in the right-hand section.
Start a new line	Click to position the cursor and press Enter.
Delete text	Drag to select the text and press Delete.
Move text	Drag to select the text, and then cut and paste.
Copy text	Drag to select the text, and then copy and paste.
Change the font, size, style, or bounding shape	Select from the Font , Size , Style , or Bounding shape lists
Add a callout	Select <Callout> and click Add Key .
Add part information (to anchored text only)	a. Click the + symbol to expand Part Information. b. Select any of the following: • <Part name> • <Part Heritage> • <Parent Assembly> • <Part List> • <Part Summary> c. Do one of the following: • Click Add Key to add a placeholder for the attribute. (The placeholder is replaced by the attribute value when you click OK). • Click Add Value to add the attribute <i>value</i> as if you typed it. The part information text appears in the Preview section of the dialog box.
Add location (to anchored text only)	Include x, y, or z coordinate location information corresponding to the coordinate values of the anchor point for the text markup. a. Click the + symbol to expand Locations.

To	Do this
	b. Select a coordinate. c. Click Add Key to add a coordinate for the text markup. d. (Optional) Click Location... to customize the location information. If Include labels is selected on the Markup text dialog, the 3D anchored text shows the appropriate positive or negative value label for the x, y, and z coordinates based on whether a label is defined. You may also customize the precision and show the absolute value of the coordinates.
Add metadata attributes (to anchored text only)	The list of metadata attributes varies depending on the information associated with model. a. Click the + symbol to expand Meta Data Attributes. b. Select an attribute. c. Do one of the following: - Click Add Key to add a placeholder for the attribute. (The placeholder is replaced by the attribute value when you click OK.) - Click Add Value to add the attribute <i>value</i> as if you had typed it. The attribute text appears in the Preview section of the dialog box.

- If you are using **3D collaborative review and markup**, and you have selected an author layer in the layer list, you have the option to add **Author Profile Information**. The profile information varies depending on the information you have defined.
- (Optional) Click **Refresh** to update the text in the **Preview** section.
- (Optional) To save the new text, attributes, and mode as the default, select the **Set as default** check box.
- Click **OK**.

8. If you are adding a part list or a part summary, do the following.
 - a. Select the parts to add to the list or summary from either the **Viewing window** or from the tree in the **Project Workspace window**.

If you are creating a part list, the parts display in the list in the order in which they are selected. If you are creating a part summary, the parts display alphabetically, showing upper case part names first.

- b. Click **Finish** on the **Select Parts** dialog.

Tip:

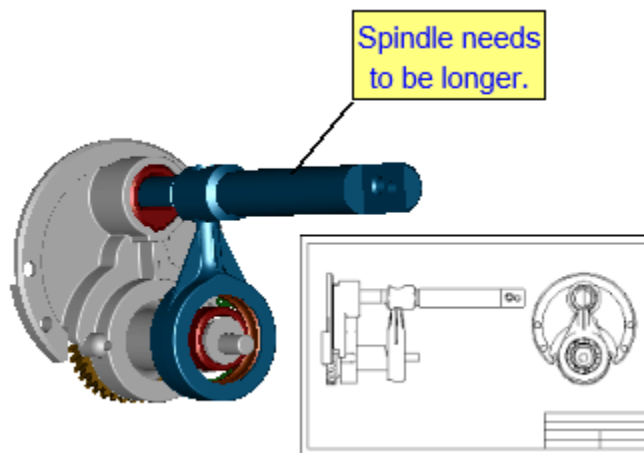
- To make Advanced mode the default, create a new text markup using Advanced mode and select the **Set as default** check box before you click **OK**.
- To clear the default text, edit the **pre-defined text** on the **Text** tab of the **Markup 3D Preferences** dialog box.

Add hyperlinks to text and inset image markups

You can make existing text and inset image markups hyperlinks in the 3D Viewing window.

Example:

For the model below, the text markup “Spindle needs to be longer.” is a hyperlink to a 2D image file, and the inset image is a hyperlink to a PDF file.



1. (If it does not exist already) Add one of the following types of markups to the Viewing window:
 - Floating inset image markup

- Anchored inset image markup
 - Floating text
 - Anchored text
2. Select the markup.
 3. Right-click the markup and choose **Properties**.
 4. (If working with a text markup) In the **Properties** dialog box, click the **Hyperlink** tab.
 5. In **Link Specification**, type the URL, or click **Browse** and browse to the appropriate file path.

You can enter a reference to any type of file in the **Link Specification** box. The application that is associated with the file type will open and display the file.

6. Click **OK**.

The letters of a text markup become blue to indicate that the markup is a hyperlink.

Note:

When you open a URL for an animation (.VAN) file, the animation plays automatically.

Tip:

To create a URL markup that opens the default email program and addresses a message, in the **Link Specification** box, type `mailto:` and the email address. When you double-click the link, the email program is launched and a message is addressed to the specified address.

7. (Optional) To navigate to the URL or launch the referenced file, double-click the link.

The item the link references appears in a separate window.

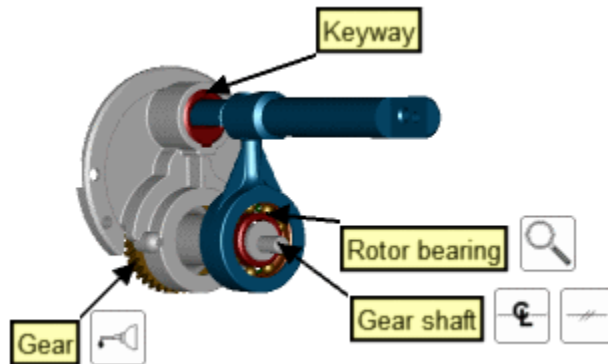
Add symbols to 3D text markups

A symbol is an image file that represents a property or operation. You can add one or more symbols to floating or anchored 3D text markups.

Example:

For the model below, you add several anchored text markups, then add some symbols to the markups. The symbols indicate that the identified parts should have additional operations

performed on them. The symbols represent “lubrication” for the gear, “a close examination” for the rotor bearing, “parallel” and “center line” for the gear shaft.



1. (Optional) **Set 3D markup symbol preferences.**
2. In the Viewing window, right-click the text markup and choose **Add Symbols**.
3. In the **Symbol Manager** dialog box, double-click a category.

All images for that category appear.

4. Select one or more images and click **Add Symbols**.

The symbols you selected are added to the text markup.

Tip:

To delete a symbol from a text markup without removing the entire text markup, right-click the symbol and select **Delete**.

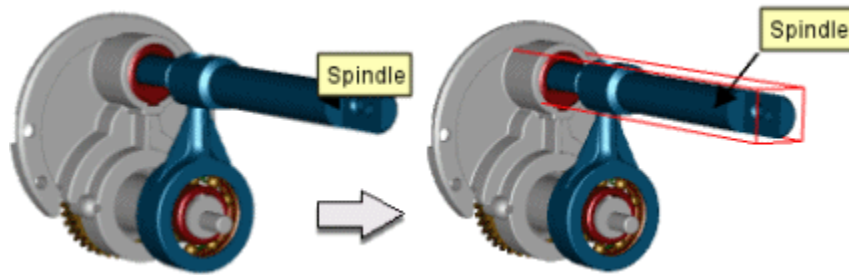
Once you add a symbol, it becomes the property of the markup it was added to.

Position anchored text markups

If you already have an anchored text markup attached to a part, you can place the markup precisely where you want it.

Example:

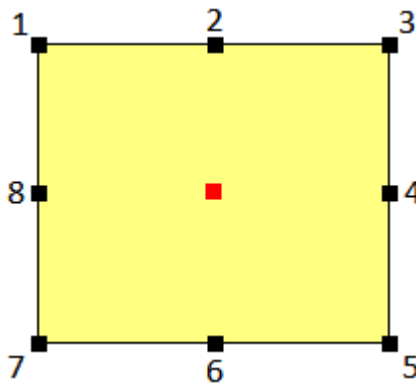
For the model below, you add an anchored text markup to identify the spindle of a product. You then use the position anchored text feature to move or reposition the text markup a specific distance away from the point on the part that it is attached to.



1. Select the part with anchored text markup.
2. On the **3D Markup** toolbar, click **Position** .
3. In the **Position Anchored Text** dialog box, from **Direction**, select where you want the anchored text markup to be after it is positioned.

Example:

In the graphic below, the red dot in the center represents the location of the anchor point for the text markup. The numbered points around the edges represent the locations where you can position the text relative to the anchor point.



Number	Position	Number	Position
1	Top left	2	Top
3	Top right	4	Right
5	Bottom right	6	Bottom
7	Bottom left	8	Left

- In **Distance**, type the number of units long you want the leader for the anchored text to be.

Note:

To have units other than pixels, you need to enable **page extents**.

- Click **OK**.

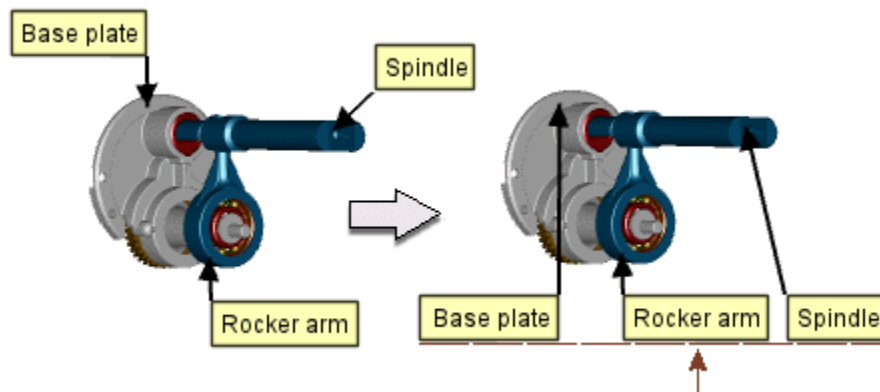
The text markup moves to the new location you specified.

Align anchored text markups

When you have multiple anchored text markups, it can be useful to align them for readability.

Example:

When using bottom alignment, the three anchored text markups move from their original position to align with the bottom-most markup.



Note:

This feature is available only if there are two or more anchored text markups on the currently selected parts.

- Select the parts with the anchored text markups that you want to align.
- On the **3D Markup** toolbar, click **Align** .
- In the **Align Markups** dialog box, from **Vertical Alignment** and **Horizontal Alignment**, select how you want the anchored text markups to be aligned.

For vertical alignment:

Use this option	To do this
Left	Align the left side of markups to the left-most markup.
Center	Align the centers of markups to the center-most markup.
Right	Align the right side of markups to the right-most markup.

For horizontal alignment:

Use this option	To do this
Top	Align the top side of markups to the top-most markup.
Middle	Align the middles of markups to the middle-most markup.
Bottom	Align the bottom side of markups to the bottom-most markup.

Tip:

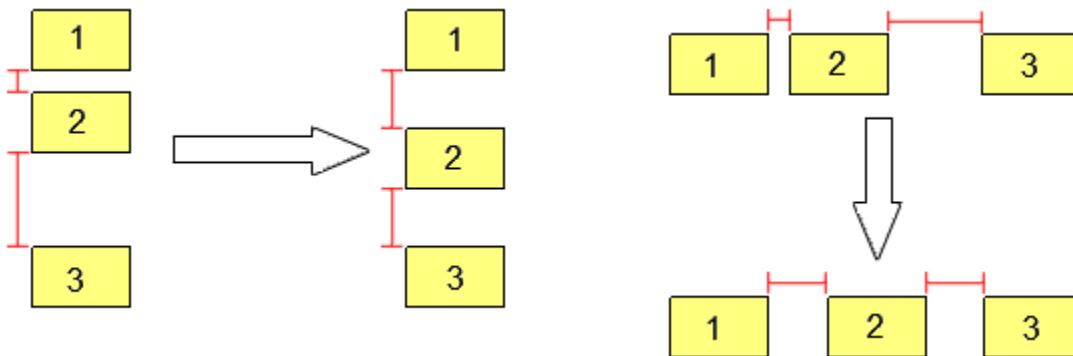
Study the original position of your markups before deciding which types of alignment to select. If you set both **Vertical Alignment** and **Horizontal Alignment**, markups may become “stacked” on top of each other and hard to read.

- Click **OK**.

The anchored text markups for the selected parts are moved to align with each other.

Distribute anchored text markups

The distribution feature determines the vertical and horizontal distance between each markup and moves the markup to evenly distribute them, either vertically or horizontally.



Note:

This feature is available only if there are three or more anchored text markups on the currently selected parts.

1. Select the parts with the anchored text markups that you want to distribute.
2. On the **3D Markup** toolbar, click **Distribute** .
3. In the **Distribute Markups** dialog box, click either **Vertical** or **Horizontal**.

Use this option	To do this
Vertical	Determine the vertical distance between each markup and move the markup to evenly distribute them.
Horizontal	Determine the horizontal distance between each markup and move the markup to evenly distribute them.

Tip:

Study the original position of your markups before deciding which types of distribution to select. If all markups are overlapping, distributing them will not make them readable. You may need to manually move two markups to get the rest distributed evenly between the two.

4. Click **OK**.

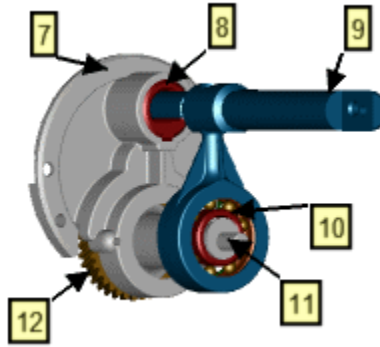
The anchored text markups for the selected parts are moved to align with each other.

Resequence callouts

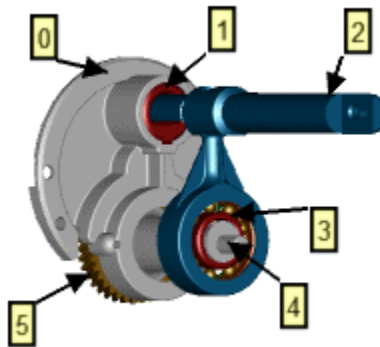
A callout is a type of text markup in which you specify letters or numbers to identify parts in a specific order on a 3D model. If you have added multiple text markups that contain callouts, you may want to resequence the callouts to start from a specific number or letter, and to not skip numbering for callouts that have been deleted.

Example:

For the model below, you add several anchored text markups that use the `<Callout>` text option to number parts for quick reference in a 3D model. Later, you delete some of the markups that you decide you did not need after all.



Rather than have the remaining text markups start with number 7 and skip some numbers, you resequence the callout numbering to restart at 0 again.



1. On the **3D Markup** toolbar, click **Resequence Callouts** .
2. In the **Resequence Callouts** dialog box, select the callout you want to edit and do any of the following:

To do this	Do this
Delete a callout	Click Delete .
Change the order of callouts in the list	Click Move Up or Move Down to move the callout to a new position in the list.
Change the value of an individual callout in the list	a. Click Resequence. b. In the Resequence Settings dialog box, under Callout Format, indicate whether the callout format should be alphabetic or numeric. c. In Identifier Starting Value, type a new number or letter for the callout.

To do this	Do this
	d. Select or clear Reset next available identifier to increment future callouts from the value specified in Identifier Starting Value. e. Click OK.
Sequentially renumber or re-alphabetize more than one callout in the list	a. From the list, select the callouts you want to renumber. Tip: To select multiple callouts, hold Shift or Ctrl while selecting callouts to select a range of callouts or individual callouts, respectively. b. Click Resequence. c. In the Resequence Settings dialog box, under Callout Format, indicate whether the callout format should be alphabetic or numeric. d. In Identifier Starting Value, type the starting value you want the selected callouts to renumbered from. e. Select or clear Reset next available identifier to increment future callouts from the last callout selected for resequencing. Example: You are resequencing 4 numeric callouts, and you specify a value of 4 for Identifier Starting Value. The callouts are resequenced as 4, 5, 6, and 7. If you selected Reset next available identifier, the next callout you add to the model will be numbered 8. f. Click OK.

Note:

You can detect duplication of callout numbers or letters by enabling collision detection in the **callouts preferences**.

3. Click **Close**.

Your changes are applied to the text markups.

Manipulating 3D markups

Select a 3D markup

1. On the **3D Markup** toolbar, click **Enable Markup** .
2. On the **3D Markup** toolbar, click **Select** .
3. Click the markup.

The markup is selected in the **Markup Layers** list and in the Viewing window. On graphic markups, boxes appear at each corner and the middle of each side of the bounding box that contains the markup.

Move 3D markups

1. Select the markup.
2. Move the markup in one of these ways:

To	Do this
Move the markup to a specific location.	In the Viewing window, drag the markup.
Place the markup on top of all the other markups on your image	Right-click and choose Bring to Front .
Place the markup behind all the other markups on your image	Right-click and choose Send to Back .

Copy text in text markups

Copy text in text markups

Right-click the text markup and choose **Copy Text**.

The text is copied to the clipboard, and ready to paste into any application that accepts pasted text.

Edit text in text markups

1. Select the text markup.
2. Right-click the markup and choose **Edit Text**.

3. In the **Edit Text** dialog box, edit the text using **Basic** or **Advanced** mode.
4. Click **OK**.

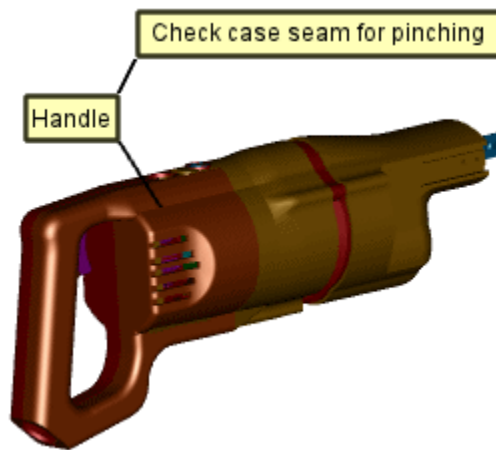
The text changes in the markup.

Add a text markup to an existing text markup

You can add a text markup to an existing text markup.

Example:

In the example below, the first text markup says *Handle*, and the text markup saying *Check case seam for pinching*. has been added to the *Handle* text markup.



The second text markup is attached to the first text markup by a leader and can be repositioned with respect to the first text markup. If you reposition the first text markup, the attached text markup moves with it.

The second text markup inherits the attributes of the first text markup. For example, if you are adding a text markup to an anchored text markup, the second text markup is automatically an anchored text markup. You can only attach one additional text markup to an existing text markup.

1. Right-click the text markup (that will have an additional text markup attached to it) and choose **Add Text Markup**.
2. In the **Edit Text** dialog box, add the text for the second markup using **Basic** or **Advanced** mode.
3. Click **OK**.

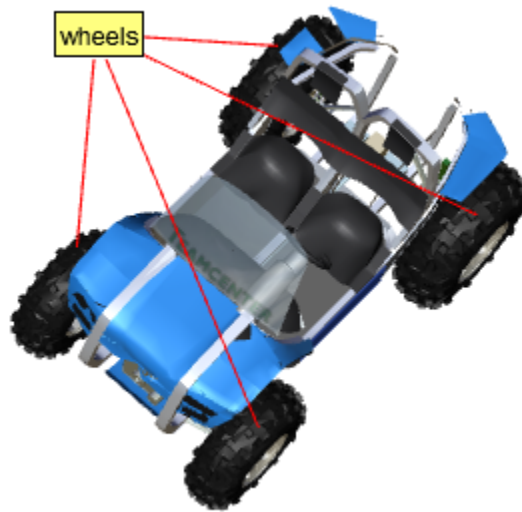
The second text markup appears near the first text markup, with a short leader attaching it to the first text markup.

Add leaders to text and inset image 3D markups

You can add additional leaders to anchored text and inset image markups. This feature is useful when you want to anchor the same markup to multiple parts, or when you want to point to multiple places on the same part.

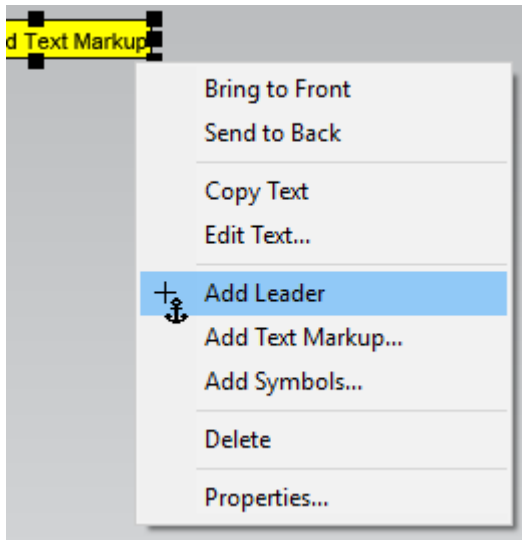
Example:

A model of a car contains four wheels. You create an anchored text markup that points to one of the wheels, then add three more leaders from the markup to the other wheels, instead of creating three more individual markups.



1. Right-click an anchored text or inset image markup and choose **Add Leader**.

The add leader mode is enabled.



2. Click the next part, or another point on the same part, to which you want to add a leader.

A leader is added to the markup.

3. Disable the add leader mode by right-clicking the text markup again and choosing **Add Leader**. The add leader mode is now toggled off.

Add hyperlinks to text and inset image 3D markups

You can link a web page or file to text and inset image 3D markups.

1. **Select** the text markup or inset image.
2. Right-click the markup, and choose **Properties**.
3. In the **Text Properties** or **Inset Image Properties** dialog box, click the **Hyperlink** tab.
4. Do any of the following:
 - To link to a web page, type the URL.
 - To link to a file, click **Browse** and browse to a file.
 - To remove a hyperlink, delete the **Link Specification** value.
5. Click **OK**.
6. Double-click the markup to open the hyperlink.

Resize 3D markups

You can resize line, rectangle, polygon, ellipse, and inset image markups.

1. Select the markup.

A bounding box appears around the markup, with square handles at the corners and midpoints.

2. Click on a handle and drag it.

To maintain the aspect ratio of inset image markups as you resize, drag a corner handle.

Note:

Inset image markups that are automatically sized cannot be resized by dragging a size handle. You can resize them using these methods:

- Edit their properties to specify new automatic **Width** and **Height** values.
- Edit their properties to set them to size manually, after which you can resize them by dragging a handle.

Delete 3D markups

To delete	Do this
Selected 3D markups	1. Select the markup. 2. Do one of the following: • Right-click and choose Delete. • Press the Delete key on the keyboard.
All 3D markups	On the 3D Markup toolbar, click Remove All  .
The last markup you added	Press Ctrl+Z. Note: To reverse an undo, press Ctrl+Y.

Display anchored markups with parts

When you turn parts on and off, markups anchored to the parts can turn on and off with them.

- On the **3D Markup** toolbar, click **View Anchored with Part** .

This mode remains activated for all markups you add after this step, until you deactivate the mode.

- To disassociate the visibility of the markups from the parts, clear **Markup**→**View Anchored with Part**.

Changing 3D markup appearance properties

Change the properties of straight line, freehand line, or polyline 3D markups

- Select** the markup to change.
- Right-click the markup, and choose **Properties**.
- In the **Line Properties**, **FreeHand Line Properties**, or **Polyline Properties** dialog box, do any of the following:

Use this option	To
Line style	Specify Solid , Dash , Dot , DashDot , or DashDotDot .
Line color	Specify the line color.
Thickness	Specify the line thickness.
Units	Specify the units used for line thickness when page extents are active.
(Line Ends) Style	Specify whether arrowheads (or other symbols) are added to the ends of lines.
(Line Ends) Size	Specify the size of symbols added to the ends of lines.

- Click **OK**.

Change the properties of ellipse, polygon, and rectangle 3D markups

- Select** the markup to change.
- Right-click the markup, and choose **Properties**.
- In the **Ellipse Properties**, **Polygon Properties**, or **Rectangle Properties** dialog box, do any of the following:

On this tab	Use this option	To
Edge	Edge style	Specify the edge style.
	Edge color	Specify the edge color.
	Thickness	Specify the edge thickness.
	Units	Specify the units used for the edge thickness when page extents are active.
Fill	Fill Enable	Add or remove fill.
	Fill style	Specify a line pattern to fill the closed shape.
	Fill color	Specify the color used to fill the closed shape.
	Transparency	Specify the level of fill transparency.

- Click **OK**.

Change the properties of 3D text markups

- Select** the text markup to change.
- Right-click the markup and choose **Properties**.
- In the **Text Properties** dialog box, do any of the following:

On this tab	Use this option	To
Line (This tab only appears for <i>anchored</i> text markups)	Line style	Specify the style of leader.
	Line color	Specify the color of leaders.
	Thickness	Specify the thickness of leaders.
	Units	Specify the units used for line thickness when page extents are active.
	Line Ends	
	Style	Specify whether to add arrowheads (or other symbols) to the Begin and End part of leaders.
	Size	Specify the size of symbols added to the Begin and End part of leaders.
	Leader Line	

On this tab	Use this option	To
	Attachments	Specify how leaders must attach to anchored text or inset image markups. Choose from the following options: • Edge Midpoint • Vertex • Center Note: If you choose a speech bubble as the bounding shape for your text markup, the leader attachment options have no effect. If you choose a circle as the bounding shape for your text markup, Center is the only leader attachment option available.
	Display on Top	Specify whether leaders must always be drawn on top of the model in the 3D view. Choose this option to retain the leader position on top when the view is rotated.
Edge	Edge style	Specify the line style used for text bounding shapes. Options are Solid , Dash , Dot , DashDot , and DashDotDot .
	Edge color	Specify the color of text bounding shapes.
	Thickness	Specify the thickness of text bounding shapes.
	Units	Specify the units used for the thickness of text bounding shapes when page extents are active.
Fill	Fill style	Specify a line pattern to fill text bounding shapes.
	Fill color	Specify the color used to fill text bounding shapes.
	Transparency	Specify the level of fill transparency.
Font	Font	Specify the font to use.
	Font Style	Specify Regular , Bold , Bold Italic , or Italic . (Some fonts have only Regular style.)
	Size	Specify the font size.
	Effect	Specify Underline or Strikeout .

On this tab	Use this option	To
	Color	Specify the text color.
Hyperlink	Link Specification	Add or edit a hyperlink.
Text	(Text Alignment) Horizontal	Specify Left , Center , or Right alignment.
	(Text Alignment) Vertical	Specify Top , Center , or Bottom alignment.
	Bounding Shape	Specify None , Rectangle , Circle , or Speech Bubble .
	Bounding Shape Size	Specify the size of the text bounding shape when page extents are active.
	(Anchored Text) Zoom with view	Specify whether to scale anchored text markups when the model is scaled, for example, when using zoom options.
	(Anchored Text) Visible when anchored parts are not found	Specify whether to make anchored text visible (upon loading a session or PLM XML file) when it is anchored to parts that have been removed from the model or moved to a different assembly in the same model. When anchored text is visible for missing parts, leader lines do not appear and the text appears in red.
Symbols	Symbol Location	Specify location relative to the text. Options are Top , Bottom , Left , or Right .
	Stacking Symbols	Specify whether symbols display in one horizontal row or in <i>stacked</i> rows.
	Symbols Per Row	Specify the number of symbols in each row if symbol stacking is active.
	(Symbol Size) Width	Specify symbol width.
	(Symbol Size) Height	Specify symbol height.
	Spacing	Specify the number of units between the text and symbols.
	Units	Specify the units used for symbol width, height, and spacing when page extents are active.
	Reorder Symbols	Change the order or delete symbols.

4. Click **OK**.

Change the properties of inset image markups

1. **Select** the inset image to change.
2. Right-click the image and choose **Properties**.
3. In the **Inset Image Properties** dialog box, do any of the following:

On this tab	Use this option	To
Line (This tab only appears for <i>anchored</i> markups)	Line style	Specify the style of leader.
	Line color	Specify the color of leaders.
	Thickness	Specify the thickness of leaders.
	Units	Specify the units used for line thickness when page extents are active.
	Line Ends	
	Style	Specify whether to add arrowheads (or other symbols) to the Begin and End part of leaders.
	Size	Specify the size of symbols added to the Begin and End part of leaders.
	Leader Line	
	Attachments	Specify how leaders must attach to anchored text or inset image markups. Choose from the following options: • Edge Midpoint • Vertex • Center Note: If you choose a speech bubble as the bounding shape for your text markup, the leader attachment options have no effect. If you choose a circle as the bounding shape for your text markup, Center is the only leader attachment option available.
	Display on Top	Specify whether leaders must always be drawn on top of the model in the 3D view.

On this tab	Use this option	To
		Choose this option to retain the leader position on top when the view is rotated.
Hyperlink	Link Specification	Add or edit a hyperlink.
Inset Image	Size manually	Make the image resizable by means of dragging grips.
	Size automatically	
	Original size	Resize the image back to its original size.
	Width and Height (editable and applied when Size manually and Size automatically are not selected)	Size the image to the specified fixed size. The Units value is set to Pixels unless page extents are enabled. If page extents have been enabled, then ensure that an appropriate units type is selected.

- Click **OK**.

Change the properties of filtered markups

You can edit properties of existing markups, including lines, shapes, text, and inset images, to standardize their appearance.

Use the following procedure to filter existing markups and apply, in one operation, specified property values to all the markups that are included in the filtered set.

- For changes to properties of anchored markup, if you intend to limit changes to only markup that is anchored to selected or visible parts, then select or make visible those parts.
- On the **3D Markup** toolbar, click **Enable Markup** .
- On the **3D Markup** toolbar, click **Edit Properties** .
- In the first **Edit Properties** dialog box, select the filter criteria (both **Types** and **Modes** sections) for creating the set of existing markups to update, and then click **OK**.

Edit Properties

Select the filter criteria to determine for which existing markups the property updates will apply.

Types

- ☐ Freehand Line
- ☐ Line
- ☐ Polyline
- ☐ Ellipse
- ☐ Polygon
- ☐ Rectangle
- ☐ Inset Image
- ☒ Text

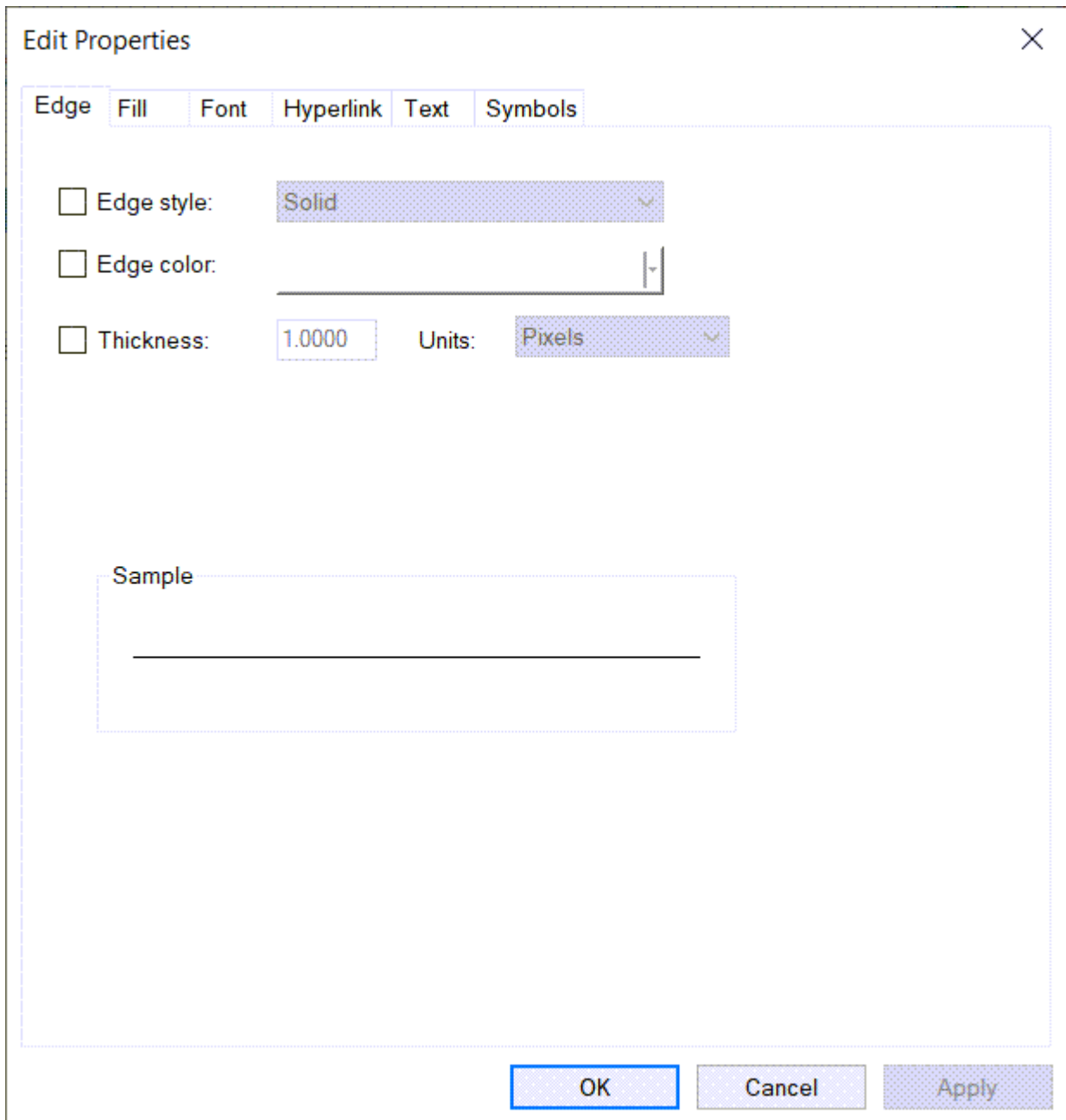
Modes

- ☐ Anchored

On:
- ☐ Unanchored
- ☒ In-Plane

For changes to properties of anchored markup, if you intend to limit changes to only markup that is anchored to selected or visible parts, then select **Anchored** and from the **On** list, select the option you want.

5. In the second **Edit Properties** dialog box, select the properties to update and the property values to apply.



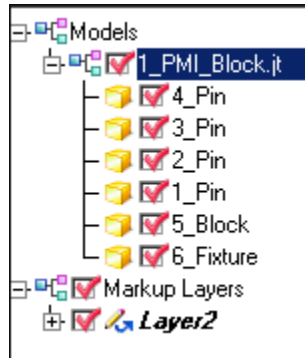
6. Click **OK**.

Working with 3D layers

Overview of working with 3D layers

When you open a 3D model, file details appear in the **Assembly** view and the model appears on the Viewing window. The file structure is reflected in a hierarchical outline. When you add markups or insert other 3D files to the base model, you are adding these elements to new markup layers that are associated with the base model.

Visual references in the **Assembly** view indicate the current visibility state of the 3D model or markup layers.



Create a new 3D markup layer

New markup layers are automatically named Layer1, Layer2 in the **Assembly** view, then saved to the markup dataset as markupName_Layer1, markupName_Layer2, and so forth.

If you have **inserted 3D markup layers** into the same base document and then added new layers, the new layers are saved to the first-opened markup dataset.

Note:

If you rename new markup layers before saving them, they are still saved as markupName_Layer#.

You can **change the name of markup layers** after the initial save.

- On the **3D Markup** toolbar, click **New Layer** .

The new layer is unmanaged until you save it as a new dataset.

Note:

A new markup layer is automatically created if you add a markup to the model using one of the markup tools.

When you create or edit a 3D layer, a pen and arrow icon  appears before the name of the layer in the **Markup Layers** list. When you save the layer, the icon disappears until the layer is edited again.

Insert a 3D markup layer

You can open one or more markup datasets in the same view for an already-open base document.

- The base document is not inserted multiple times; only the markup layers are inserted.
 - New layers are saved to the first-opened markup dataset.
 - If you have used the same layer names in different markup datasets, then inserted the markup layers from one dataset into another dataset, the layer names in the hierarchy are the same, and you can not tell which dataset a layer is from.
 - You cannot query which dataset the layers are from, or what their properties are (owner, permissions, and so forth).
1. (If **Markup Layers** list exists) In the **Assembly** view, right-click **Markup Layers** and choose **Insert Layer**.
 2. In the **Open** dialog box, browse to and select the .vpl layer file to insert, and then click **Open**.

The new layer is treated as unmanaged until you save it. Inserted, modified layers are also treated as unmanaged. The layer is saved to its original location.

Note:

- When you right-click **Markup Layers** and choose **Insert Layer**, you can only insert a .vpl 3D markup file that is located in a local file system or on the network.

Display and hide 3D markups

You can display, hide, and rearrange 3D markups.

Do one or more of the following:

To	Do this
Display or hide individual 3D layers or 3D markups.	In the Assembly view, in the Markup Layers list, select or clear the check box to the left of the markup layer or markup name  .
Rearrange 3D markups.	In the Assembly view, in the Markup Layers list, click the markup you want to move and drag it to its new position.

Rename 3D markup layers

Note:

If you rename new markup layers before saving them, they are still saved as markupName_Layer#.

You can change the name of markup layers after the initial save.

1. In the **Markup Layers** list, right-click the markup layer or an individual markup and choose **Rename**.

The name highlights, and an edit box appears around the name.

2. Type the new name.

Delete markup layers and markups

Select the markup layer or the markup and delete it in one of these ways:

- In the **Assembly** view, right-click the layer name and choose **Delete Layer**.
- In the **Assembly** view, right-click markup name and choose **Delete**.
- In the Viewing window, right-click the **selected** markup and choose **Delete**.

View markup layer properties

In the **Markup Layers** list, right-click the layer name and choose **Properties**.

The **Layer Properties** dialog box displays the dataset name, parent document ID, layer owner (creator) and owner group, layer creation/modification time, checked-in or checked-out status, and read/write permissions.

Save 3D markup layers

1. In the **Assembly** view, select the layer you want to save, right-click and choose a save option from the list.
2. Choose one of these options:

Choose this option	To save
Save All Layers	All markup layers associated with the base document. Note: Modified markup layers are saved to their associated markup datasets. New layers are saved to a new markup dataset.
Save Selected Layer	Highlighted markup layer associated with the base document.
Save Selected Layer As	Highlighted markup layer as a new markup dataset associated with the base document.

Under certain conditions, warning messages display when saving legacy layer files (release 12.4 and earlier). To avoid overwriting the legacy layer file, which will cause issues with legacy viewers, on the warning dialog click **Save as** and choose a different file location to create a new layer file.

3. In the **Save** dialog box, select or type the name of the markup layer file to save.

Note:

You cannot use any non-ASCII character, including 8 bit accented Western European and multi-byte characters, for a file name.

4. Click **Save**.

(If you chose **Save All Layers**) A new **Save** dialog box appears for each markup that you save.

Note:

When you create or edit a 3D layer, a pen and arrow icon  appears before the name of the markup layer in the **Markup Layers** list until you save the layer. When you save the layer, the icon disappears until the layer is edited again.

Export 3D markup layers

1. In the **Assembly** view, right-click the layer name and choose **Export Layer**.
2. In the **Save Layer (layer_name) As** dialog box, accept the default name or type a new layer name. You can then accept the default local file system location or navigate to a new folder.
3. Click **Save**.

The markup layer is saved with a .vpl extension.

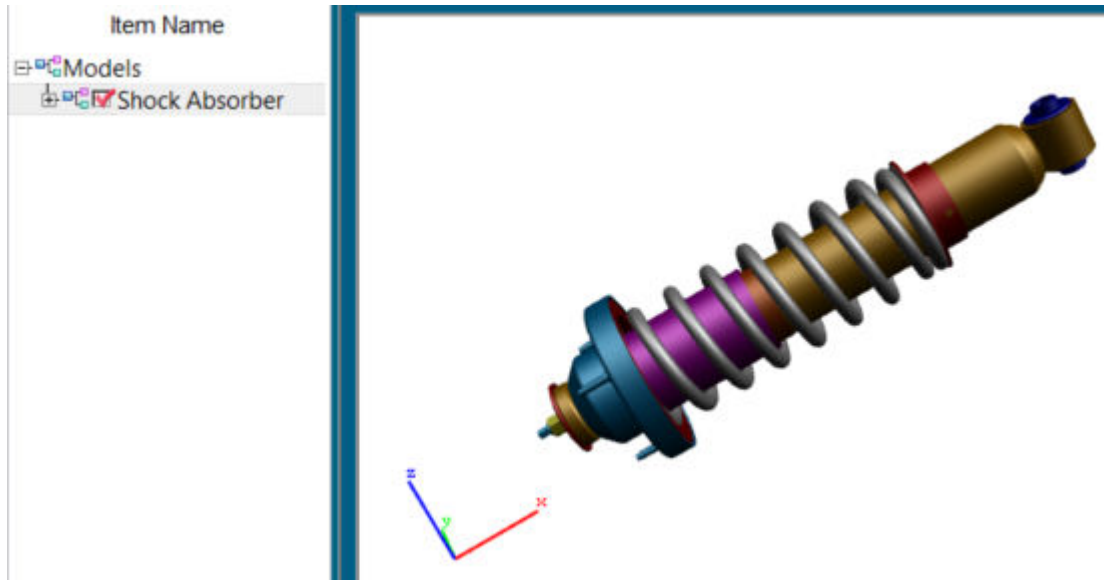
Note:

- The benefit of saving export layers generated from the standalone application viewer is the saved data can then be reused more easily with other team members.
- Saving export layers is not supported by Teamcenter.

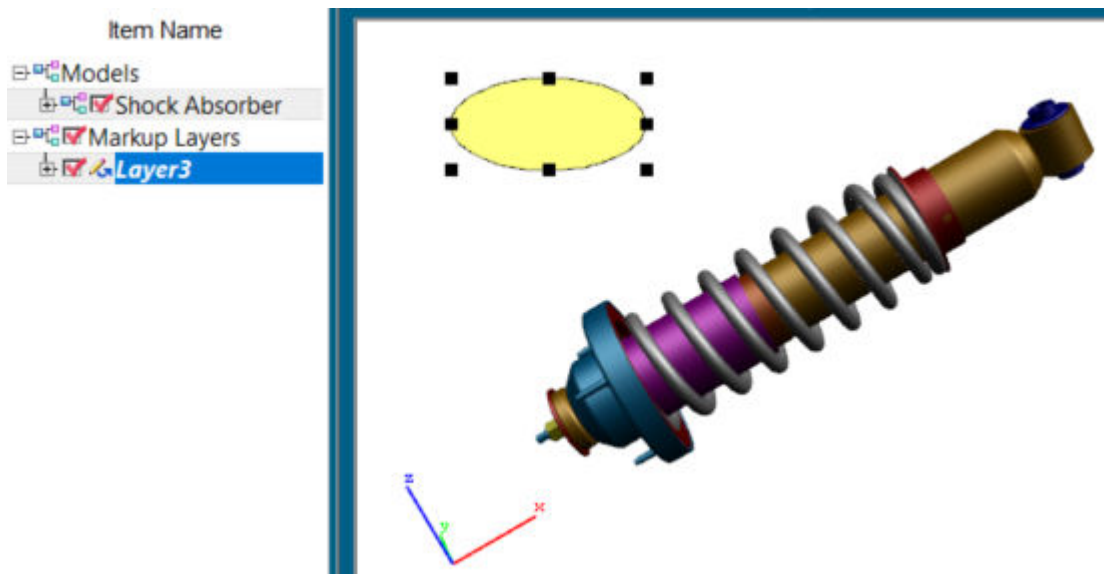
Example:

For this example, use *shock_absorber.jt* which is saved in **User Areas→Examples→3D**.

1. From the standalone application viewer, open a 3D model.



2. Add a markup layer to the base model.



3. In the **Save Layer** (*layer_name*) **As** dialog box, accept the default name and location or change the layer name and its location.

Tip:

Because reloading export layers is a two step process, It is recommended that you save export markup layers in the same location as the base 3D model.

4. Click **Save**.

The markup layer is saved with a .vpl extension. This example saves the .vpl file as *Test.vpl* and in the same local file system location, **User Areas→Examples→3D**.

Now you can choose **File→Open** and navigate to the location where the base 3D model and export layer are saved. Open the base model, *shock_absorber.jt* and then open *Test.vpl*.

The base 3D model and markup layer are displayed in the Viewing window.

Copy or move 3D markups

You can move or copy a markup to a different markup layer.

1. In the **Markup Layers** list, right-click the markup and choose **Cut** to remove the markup from the original layer, or **Copy** to leave the markup on the original layer.
2. Right-click the destination markup layer and choose **Paste**.

The markup appears in its new location in the **Markup Layers** list. The markup is also now on the new markup layer in the Viewing window.

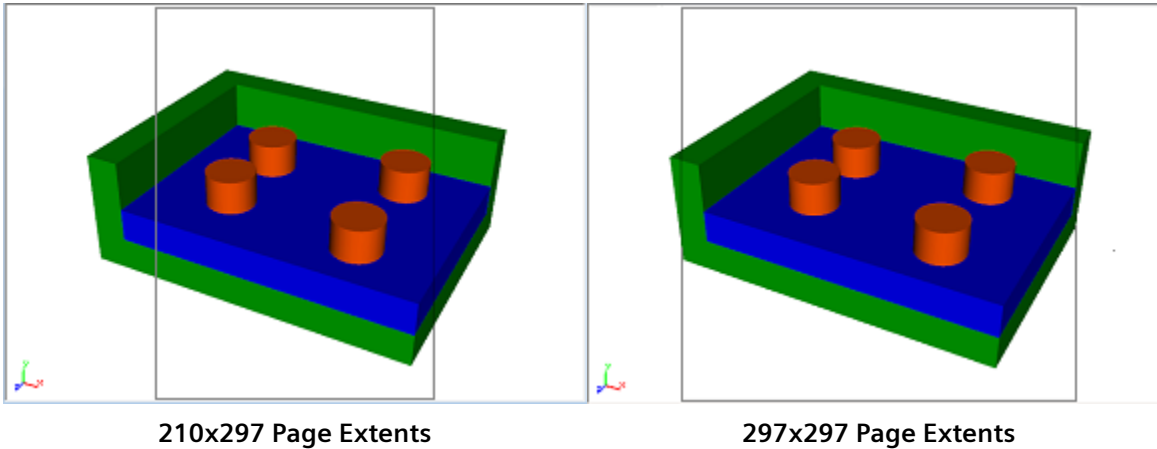
Note:

If you copy a markup, the markup is pasted over the original markup in the Viewing window. It may seem that nothing happened when you clicked **Paste**. To see the new markup, select and drag the pasted markup to another location in the Viewing window. The original markup remains in its original location.

Page extents

Overview of page extents

Page extents define a page size (width and height) for viewing your 3D model. When you enable page extents, print output is limited to what is within the page extents rather than what is in the Viewing window extents.



- You can define multiple page extents and choose which one to use.
- You can display the page extents *visual guide* rectangle in the Viewing window.
- 3D markup size properties, like line thickness and font size, automatically scale relative to the page extents.
- When you view **JT files containing watermarks**, the watermarks automatically adjust to fit within the page extents.

Note:

You may notice degradation in the quality of markups while using page extents. The degraded quality of any markups as seen in the 3D view when a page extent is active is not reflected in the ultimate 2D output produced from these features.

Enable page extents

1. Choose **View**→**Page Extents**.
2. In the **Page Extents** dialog box, select the **Enable Page Extents** check box.
3. (Optional) Add a new page extents or edit an existing one.
4. Click the name of the page extents to use.
5. (Optional) Select the **View Visual Guide** check box to display a rectangle showing the page extents in the Viewing window.
6. Click **OK**.

Note:

Once enabled, the page extents feature is active for all work sessions. To turn off page extents, choose **View→Page Extents** and clear the **Enable Page Extents** check box.

Define new page extents

1. Choose **View→Page Extents**.
2. In the **Page Extents** dialog box, click **Add**.
3. In the **Add Page Extents** dialog box, enter a **Name** for the page extent set.
4. Chose the **Units**.
5. Enter **Width** and **Height** values.
6. Click **OK**.

The new page extents appears in the list and is selected.

Edit page extents settings

1. Choose **View→Page Extents**.
2. In the **Page Extents** dialog box, click the name of the page extents to modify.
3. Click **Edit**.
4. In the **Edit Page Extents** dialog box, change any of the following:
 - **Name**
 - **Width**
 - **Height**
 - **Units**
5. Click **OK**.

The page extents set is updated with your changes.

Delete page extents

1. Choose **View→Page Extents**.
2. In the **Page Extents** dialog box, click the name of the page extents to delete.
3. Click **Delete**.
4. Click **Yes**.

The page extents set you selected is deleted.

Change the color of the page extents visual guide

1. Choose **View→Page Extents**.
2. In the **Page Extents** dialog box, select the **Enable Page Extents** check box.
3. Select the **View Visual Guide** check box.
4. Click **Customize**.
5. In the **Customize Visual Guide** dialog box, select color.
6. Click **OK**.

The visual guide changes to the color you specified.

Managing symbol libraries

Open the symbol library

From the **3D Markup** toolbar, click **Manage Symbol Library** .

The **Symbol Manager** dialog box appears.

Delete symbols

1. **Display the layer for the text markup** containing the symbol you want to delete.
2. Right-click the symbol you want to delete, and then choose **Delete**.

The symbol is deleted from the text markup it is part of.

Create categories in the symbol library

1. **Open** the symbol library.
2. Select **New Category**.
3. In the **Symbol Manager - Create New Category** dialog box, type a name in **Category Name**.
4. Click **Browse**.
5. In the **Import Image into Symbol Library** dialog box, browse to an image file and click **Open**.
6. Click **OK**.

A new category is created, illustrated by the image file you chose.

Delete categories from the symbol library

1. **Open** the symbol library.
2. Select the category you want to delete.
3. Click **Delete**.

The selected category is deleted from the library.

Note:

The images in the category are not deleted.

Import images into the symbol library

1. **Open** the symbol library.
2. Double-click a category in which you want to add symbols.
3. Click **Import Symbols**.
4. In the **Import Image into Symbol Library** dialog box, do one of the following:
 - Select an image file.
 - Hold the Shift key and select multiple image files.
5. Click **Open**.

Note:

- You can import .JPG, .BMP, .GIF, .TIF, .PNG raster files, and .CGM vector files into the symbol library.
- If you import a .CGM vector file, when the CGM symbol appears in the 3D scene, it will be rendered to a bitmap.

The images appear in the category you selected.

Import libraries into the symbol library

You can import an entire directory of images into your symbol library; each sub-directory creates a new category within the library. If you create this directory structure on a Teamcenter server, you can use the Symbol Manager to access the centralized repository of files. Symbol library files that you may have on your local machine cannot be stored on the Teamcenter server because there is not a dataset for them.

1. **Open** the symbol library.
2. Click **Import Library**.
3. In the **Import Library into Symbol Library** dialog box, navigate to and select the folder of symbol images you want to add.
4. Click **Open**.

Note:

- The folder can contain .jpg, .bmp, .gif, .tif, .png raster files, and .cgm vector files for the symbol library.
- .cgm vector files are rendered to a bitmap when the CGM symbol appears in the 3D scene.

A new category for the library you are importing appears in the **Symbol Manager** dialog box.

Working with 3D collaborative review and markup

Overview of working with 3D collaborative review and markup

With 3D collaborative review and markup, you can collaborate with others to add or modify markups to a 3D view. Each user's markups are confined to specific layers and distinguished from other markups by font and fill attributes.

Use collaborative review and markup to perform the following tasks:

- Understanding authors
- Understanding authoring sessions
- Configuring authors
- Setting author attribute and profile information
- Using layers for 3D collaborative markup

Overview of the author's role

Authors can collaborate to add or modify markups on a 3D view. Each author owns the markup they add to the 3D view and it is visually distinctive from other markups. There are two types of authors: authenticated authors and pseudo authors.

Authenticated authors are authenticated against a server and are the only ones who can edit their profile information.

Pseudo-authors can associate font and fill attributes with a layer without having to authenticate. Therefore, the attribute and profile information of pseudo-authors is associated with a layer. Generally, you can use pseudo-authors when there is no authentication server, or for a *common* layer that no one owns.

Note:

- Anyone can edit the profile information for a pseudo author.
- Layers created by a pseudo-author are non-controlled and edited by anyone.
- You cannot change the status of pseudo-author layers to controlled. Controlled layers can only be edited by the associated author.

Authoring sessions

Overview of authoring sessions

In the collaboration atmosphere of review and markup, multiple authors together in a room with a single computer. One author (the Facilitator) makes changes based on input from the others, using a **single room with a single computer**.

Single room, single computer scenarios

To create markups under	All authors are located in a single room, the Facilitator operates the computer and creates markups under instruction from the authors.

instruction of other authors	1. Individual authors authenticate themselves. For each authenticated author, an entry appears in the Author Legend dialog box. 2. An author instructs the Facilitator to create a markup for him. 3. The Facilitator selects the author's entry from the Author Legend dialog box and creates the markup. 4. If a layer does not already exist, the Facilitator creates one. 5. If the author already has a layer displayed, the Facilitator could select that layer. 6. The Facilitator creates a markup with the author's attributes. For text markups, the Facilitator could include the author's profile information in the markup.
To switch between authors	1. An author asks the Facilitator to create markups. 2. A second author asks the Facilitator to create markups for him. 3. The Facilitator selects the second author's entry from the Author Legend dialog box. 4. The Facilitator draws a new markup. 5. The Facilitator creates a new layer that is owned by the second author. The markup is drawn with the second author's attributes.
To modify markups	1. An author informs the Facilitator to modify a markup. 2. The Facilitator selects the markup in the scene. 3. The layer tree automatically selects the containing layer. The owner of that layer will automatically be selected in the Author Legend dialog box. 4. The Facilitator makes the requested change.
To move markups between authors	1. An author instructs the Facilitator to move a markup from his layer to another author's layer. 2. Using the layer tree, the Facilitator moves the markup from the first author's layer to the second author's layer. (Assume that it is a text markup that contains profile information from the first author.)

	3. The attributes of the markup will automatically change from the first author's attributes to the second author's attributes. The textual profile information will not change.
To save layers	1. The authoring session is finished and the Facilitator is instructed to close the application. 2. The Facilitator does one of the following: • shuts down the application • creates a session file • deletes each individual layer 3. The Facilitator is prompted to save the layer files. If the Facilitator chooses to save the files to the PDM system, the Facilitator will be prompted to login to the PDM system. 4. The layer files will be saved under the ownership of the Facilitator because it was the Facilitator who logged into the PDM system (or logged into that computer). Note: The Facilitator will not be able to edit the layer files himself in the future, because he would need to authenticate himself as the owner of the layer.

Configuring authors

Configuring authors

You can configure authors in the following ways:

- Create authors
- Authenticate authors
- Assign authors to a non-authored layer
- View authors associated with currently active authored layers
- Deauthorize authors
- Delete authors

Create authors

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, click **Create Author**.
3. In the **Author Profile Information** dialog box, enter a name for **Username**.
4. With **Full name** selected, click **Edit**.
5. In the **Edit Profile Information** dialog box, enter data in **Value** to represent the author's full name.
6. Click **OK**.
7. Select and enter data in the other required keys (Phone, Email, Title, and Company) as you did for Full Name.
8. (Optional) Click **Add** to add a new profile key.
9. (Optional) Click **Pseudo-author**.
10. Click **OK**.

The new author appears in the **Author Control** dialog box.

Authenticate authors

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, select an author from the list.
3. Click **Authenticate**.

Note:

Author information is authenticated with a Visualization Data Integration Services-enabled server. However, it is for authentication purposes only; no attribute or profile information is stored on the server.

4. In the **Author Login** dialog box, enter a **Password** for the author.
5. Select a **Server** for the author information to authenticate against.
6. Click **OK**.

Note:

To use the authentication aspect you must use a supported server. If you do not have a supported server, you can use this functionality as a pseudo user.

Assign authors to a non-authored layer

1. From the Assembly tree, right-click on a non-authored layer, and select **Authors**.
2. In the **Author List** dialog box, select an author from the list.

Note:

The Author List displays all currently authenticated authors.

3. Click **Choose**.

The author you chose becomes assigned to the layer.

View authors associated with currently active authored layers

Choose **Markup**→**Layer Authoring**→**Author Legend**.

The **Author Legend** dialog box appears.

Deauthorize authors

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, select an authenticated author.
3. Click **Sign off**.

Delete authors

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, select an author from the list.
3. Click **Delete**.

The author is deleted.

Setting author attribute and profile information

Overview of attribute and profile information

You can edit a user's **profile** to associate personal information, such as name, telephone number, or email address with that user. Anyone can view profile information, but only the associated author can edit it. You can also insert profile information into text markups.

You can also set user **attributes**, which include font and fill color to markups to distinguish one user's markups from another. Attribute information can be viewed and edited by any author.

Note:

- The author attribute and profile information is persisted to the local machine's registry, not to the server. You can export and copy this information to other machines.
- It is possible that author attribute information may conflict. For example, two authors may choose the same font and fill color. To automatically avoid conflicts, use **Resolve Conflicts**.

Edit author profile information

Note:

Only authors can edit their own profiles (and only if they are **authenticated**), but anyone can view author profiles.

1. From the Assembly tree, right-click the authored layer, and select **Edit Profile**.
2. (Optional) In the **Author Profile Information** dialog box, click **Add** to add a new key to the profile.

Note:

- If the author is authenticated, you can edit the profile information. If the author is not authenticated, the dialog will appear as read-only.
- When you change an author's profile information, the changes are not reflected in existing markups. Only new text markups will reflect the profile changes.

3. (Optional) Select a key and click **Edit** to edit an entry.
4. (Optional) Select a key and click **Delete** to delete an entry.

Note:

You cannot delete fixed keys. Fixed keys include: name, phone, email, title, and company.

The author profile information changes as you specified.

Edit author attribute information

1. From the Assembly tree, right-click the authored layer, and select **Edit Attributes**.
2. In the **Author Attributes** dialog box, edit any of the following author attributes:

Note:

When you change an author's attribute information, the changes are reflected immediately in existing markups.

On this page	Use this option	To do this
Line	Line Style	Change the style of markup lines.
	Line Color	Change the color of markup lines.
	Thickness	Specify the thickness of lines. Type a larger number to draw a thicker line, or type a smaller number to draw a thinner line.
	Style	Change the style of the ends of lines. a. Choose a style in Begin for the beginning of your lines (where you click initially). b. Choose a style in End for the end of your lines (where you release the mouse).
	Size	Change the size of the ends of lines. a. Choose a size in Begin for the beginning of your lines. b. Choose a style in End for the end of your lines.
Edge	Edge Style	Change the style of the markup borders. Note: Anchored text markups that have an edge thickness greater than one are limited to the Solid style.
	Edge Color	Change the color of markup borders.
	Thickness	Change the thickness of borders.

On this page	Use this option	To do this
		Type a larger number to draw a thicker edge, or type a smaller number to draw a thinner edge.
Fill	Fill Style	Change the style of filled markups.
	Fill Color	Change the color of filled markups.
	Transparency	Change the transparency of filled markups. • Drag the slider to the left to make the fill more transparent. • Drag the slider to the right to make the fill more opaque.
Font	Font	Change the markup font in text markups.
	Font Style	Change the style of font in text markups.
	Size	Change the font size for text markups.
	Underline	Underline text in text markups.
	Strikeout	Use strikeout style in text markups.
	Color	Change the font color in text markups.

3. Click **OK**.

The author attribute information changes as you specified.

Automatically resolve attribute information conflicts

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, click **Resolve Conflicts**.
3. (Optional) In the **Resolve Conflicts** dialog box, select **Font Color** to indicate that the font should be unique to the user.

Note:

Resolve Conflicts automatically reassigns attributes for all authors, not just those that conflict.

4. (Optional) Select **Fill Color** to indicate that the fill color should be unique to the user.
5. Click **OK**.

The conflict is resolved.

Import or export attribute and profile information

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, click **Import** or **Export**.
3. Do one of the following:
 - (To import) In the **Select Author File to Import** dialog box, browse to an xml file and click **Open**.
 - (To export) In the **Select Author File to Export to** dialog box, specify a destination for the xml file and click **Save**.

The information is either imported or exported.

Using layers for 3D collaborative markup

Overview of layers for 3D collaborative markup

With 3D collaborative markup, you can collaborate with other authors to apply markups to the same 3D view. Each author's markup is saved to its own layer, which is an authored layer. Authored layers are identified from other authored layers by unique font and fill attributes. Non-authored layers are not associated with a particular author. All legacy layers are considered non-authored layers.

Create a new layer

Note:

The author for which the new layer is created must be authenticated for the new layer button to be enabled.

1. Choose **Markup**→**Layer Authoring**→**Author Control**.
2. In the **Author Control** dialog box, select the author who will own the new layer.
3. Click **Create New Layer**.

A new layer is created for the current 3D view, and the selected author's name appears in the tree in the **User** column.

Tip:

View and select the new layer, labeled with the author's name, from the Assembly page.

Save layers

Note:

- Authors that own the layers will appear in the Author Legend when they open the layers.
- If you open a layer file and no author information is found in the registry, the author information will be read from the layer file.

1. **Authenticate** an author.
2. **Create** a layer with markups.
3. **Save** the layer.

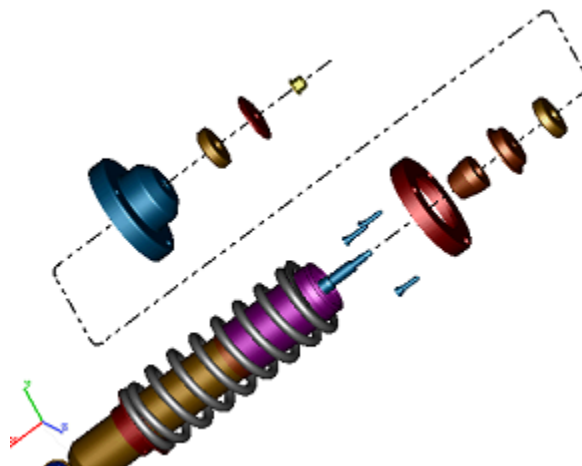
Generate reports

1. Choose **Markup**→**Layer Authoring**→**Generate Report**.
2. Use the **report wizard** to create a report.

Working with thrustlines

Thrustlines overview

Thrustlines are 3D markups that show the order and relationship of parts in assemblies.



You can

- Create thrustlines.

- Manipulate thrustlines.
- Align and associate parts with existing thrustlines.
- Perform change reconciliation on parts associated with thrustlines.
- Set thrustline appearance preferences.

Create thrustlines

1. On the **3D Thrustline Editor** toolbar, click **Thrustline** .
2. (Optional) On the **3D Thrustline Editor** toolbar, click **New Layer** .

A new layer is created and appears in the **Assembly** view.

3. On the **3D Thrustline Editor** toolbar, click **Part Feature Aligned** .
4. If necessary, set the part feature **selection** filter to the correct feature type.

Tip:

Part works well for cylindrical parts.

5. In the Viewing window click a part or part feature, which will be the beginning of the thrustline.
6. Click a second part or part feature, which represents the end of the thrustline.

The thrustline appears between the parts or part features you selected. The part or parts are associated with the new thrustline.

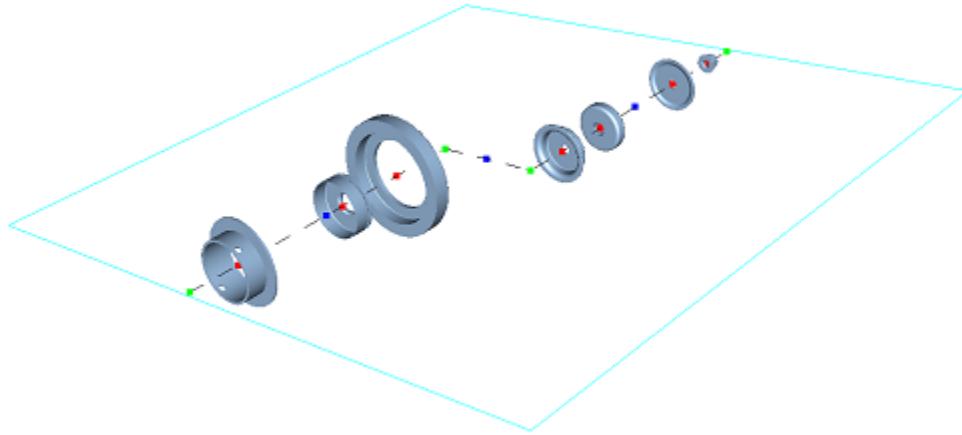
Once you create the thrustline, you can **associate** more parts with it.

Note:

New thrustlines are added to the active markup layer. If a layer is not available, a new one is automatically created.

Manipulate thrustlines

You can move, join, split, and delete thrustlines. You can also insert corners and change the appearance of thrustlines.



1. On the **3D Thrustline Editor** toolbar, click **Thrustline** .
2. In the Viewing window, select a thrustline.

The thrustline bounding area and control points appear.

3. Do any of the following:

To	Do this
Move thrustlines and their associated parts.	Do one of the following: • Drag the thrustline. • Hold Ctrl and drag a part associated with the thrustline.
Move thrustline segments and their associated parts.	Drag blue segment midpoint control points.
Lengthen and shorten thrustlines.	Drag green thrustline endpoint control points
Insert corners into thrustlines	a. Right-click the thrustline (at the point where you want to insert the corner) and choose Insert Corner. b. Drag the blue control point to manipulate the thrustline.
Join two thrustlines	a. Click the green endpoint to be the start of the joining segment. The green endpoint becomes a yellow control point. b. Right-click the thrustline and choose Join Thrustlines. c. Click the second thrustline.

To	Do this
	d. Click the green endpoint to be the end of the joining segment. The two thrustlines are joined. Note: When you join two parallel thrustlines, the result is an S-shaped thrustline. When you join two non-parallel thrustlines, the second thrustline moves into the same plane as the first thrustline, and the joining segment is a straight line.
Split thrustlines into two thrustlines	Right-click the thrustline (at the split point) and choose Split Thrustline .
Delete thrustlines	Do any of the following: - Right-click the thrustline and choose Delete. - On the 3D Thrustline Editor toolbar, click Delete Thrustline . Note: You can use the Undo command to undo the last delete action.
Modify properties of thrustlines	a. Right-click the thrustline and choose Properties. b. In the Thrustline Preferences dialog box, set new preferences.

Move parts along thrustlines

- On the **3D Thrustline Editor** toolbar, click **Thrustline** .
- Do one of the following:
 - Press Shift key and drag the associate part along the thrustline.
 - Click the thrustline and drag the associated part's red control point.

Adding parts to existing thrustlines

Overview of adding parts to existing thrustlines

When you create a new thrustline, only the part or parts used to define the thrustline are associated with it.

- When you move a thrustline, the associated part move with it.
- You can press Shift and drag to move associated parts along a thrustline

You have two options for adding other parts to existing thrustlines.

- Use **Align Part** to associate a part and move it to a point on the thrustline, or to move an already associated part to the thrustline.
- Use **Associate** to associates parts without moving them.

You can also disassociate parts from thrustlines.

Align parts with thrustlines

Move parts onto thrustlines.

1. On the **3D Thrustline Editor** toolbar, click **Thrustline** .
2. Click a thrustline.
3. Right-click the thrustline (at the location where you want to align the part) and select **Align Part**.

A red control point appears at the alignment location.

4. If necessary, set the part feature **selection** filter to the correct part feature type.
5. Click the part feature to align.

The part moves onto the thrustline. You can drag the red control-point to move the part along the thrustline.

Associate parts with thrustlines

When you create a new thrustline, at least one part is associated with it. If you create a thrustline by selecting part features on two parts, both parts are associated with the thrustline.

You can associate additional parts with existing thrustlines. You can associate parts individually, or multiple parts at once.

1. On the **3D Thrustline Editor** toolbar, click **Thrustline** .
2. Do any of the following:

To	Do this
Associate a part with a thrustline	a. Click the thrustline. b. Right-click a part, and choose Associate with Thrustline. If no control point is active, a new control point is added.
Associate a part and add it to an existing control point	a. Click the thrustline. b. Click a red control point to make it active (yellow). c. Right-click a part, and choose Associate with Thrustline.
Associate a part at a selected point along the thrustline	a. Select the part. b. Click the thrustline. c. Right-click the thrustline (at the point where you want to add the control point), and select Associate with Thrustline. A new red control point appears on the thrustline.
Associate multiple parts at a single new control point	a. Select several parts. b. Click the thrustline. c. Right-click the thrustline (at the point where you want to add the control point), and select Associate with Thrustline. A new red control point appears on the thrustline.

Disassociate parts from thrustlines

Right-click a part that is associated with the thrustline and choose **Disassociate from Thrustline**.

The part is disassociated from the thrustline.

Set thrustline preferences

Specify preferences for thrustline attributes.

1. On the **3D Thrustline Editor** toolbar, click **Thrustline** .
2. To change properties of an existing thrustline, select the thrustline, right-click the thrustline, and choose **Properties**.
3. To set preferences for new thrustlines, on the **3D Thrustline Editor** toolbar, click **Preferences** .
4. In the **Thrustline Preferences** dialog box, adjust any of the following settings:

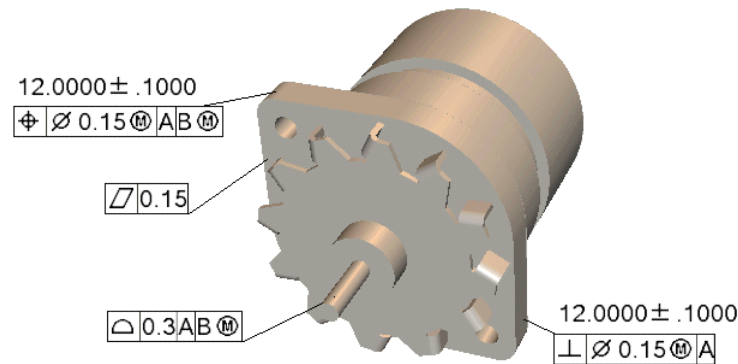
Use this option	To do this
(Line) Style	Specify line style. Note: The appearance of dashed or dotted lines in the 3D view may differ depending on the perspective of the line in the view.
Apply style to corners	Apply the line style to the corners of the line.
Color	Specify thrustline color.
Thickness	Specify the line thickness by entering a number and then selecting the measurement type (inches, picas, pixels, etc.).
Radius	Specify a number, which determines the radius of the line corners. You can specify either sharp or round corners. If you choose round corners, you can specify the radius of the curve (non-rounded corners have a radius of 0).
(Line Ends) Style	Specify the line style at the beginning and end of the line.
(Line Ends) Size	Specify the size of the line style.
Sample	Preview the line and corner specifications that you chose.

5. Click **Apply** to apply your changes.

Working with 3D GD&T markups

Overview of 3D GD&T markups

GD&T markups are text or graphical elements that you draw on a markup layer in the Viewing window. Using GD&T markups, you can add tolerance specifications to your files.



GD&T markups consist of:

Feature control frames	$\phi \varnothing 0.1 \text{ ABC}$ $\varnothing 0.025$ $\phi \varnothing 0.50 \text{ ABC}$ $\phi \varnothing 0.25 \text{ AB}$ $\varnothing 0.3 \text{ AB} \text{ M}$ $\perp \varnothing 0.15 \text{ M A}$ $\phi \varnothing 0.15 \text{ M AB} \text{ M}$
Notes	GD&T notes can include the following special symbols: $\checkmark$ $\sqcup$ ∇ $\Rightarrow$ $\triangle$ $\square$ $\leftrightarrow$ $\pm$ $^\circ$ $\varnothing$ $S\varnothing$ ST
Datums	$\perp \varnothing 0.15 \text{ M A}$ B AA C
Tolerances	$4 \times \varnothing 3.000 \pm .002 \text{ THRU}$ $90.0000 \pm .0500$ $\varnothing 2.0000 \begin{smallmatrix} +.0005 \\ -.0000 \end{smallmatrix}$

You can

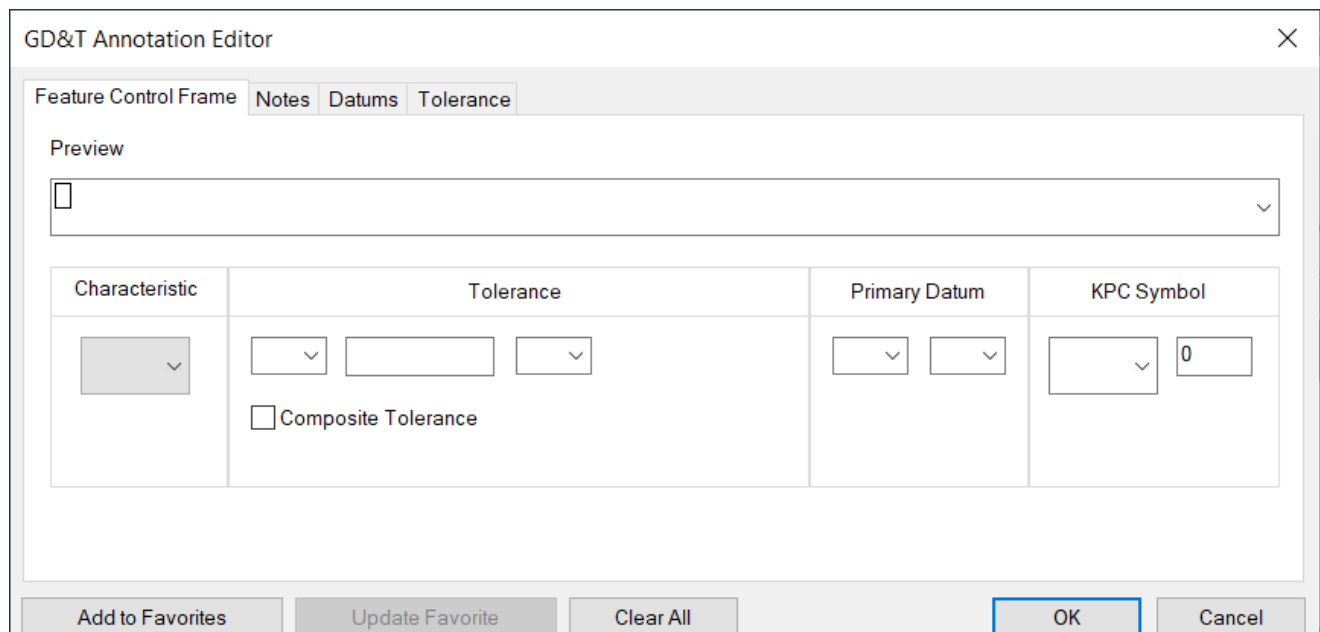
- Open the GD&T annotation editor
- Create a feature control frame
- Create GD&T notes
- Create GD&T datums

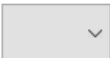
- Create GD&T tolerances
- Insert GD&T markups into 3D models
- Create a GD&T markup stack
- Edit GD&T markups
- View GD&T markups
- Change GD&T markup preferences
- Save 3D GD&T markups
- Associate 3D GD&T markups to a view

Open the GD&T annotation editor

Use the **GD&T Annotation Editor** dialog box to author feature control frames, notes, datums, and tolerance GD&T markups. The editor displays supported options.

- On the **GD&T Markup** toolbar, click **GD&T Annotation Editor** .



Characteristic	Tolerance	Primary Datum	KPC Symbol
	 	 	 0

☐ Composite Tolerance

Create GD&T feature control frames

Create GD&T feature control frames that can include geometric tolerance symbols, their values, modifiers, and datum references.

1. Do one of the following:

To	Do this
Anchor the markup to a part.	On the GD&T Markup toolbar , click Anchor Mode  .
To add the markup to a GD&T markup stack.	On the GD&T Markup toolbar , click Stack Mode  .
To place the markup anywhere in the view.	On the GD&T Markup toolbar , ensure Anchor Mode and Stack Mode are not active.

2. On the **GD&T Markup toolbar**, click **GD&T Annotation Editor** .
3. In the **GD&T Annotation Editor** dialog box, on the Feature Control Frame page, select a feature control frame characteristic from the **Characteristic** section.

Use this symbol	To represent this
	Straightness
	Flatness
	Circularity
	Cylindricity
	Profile of a line
	Profile of a surface
	Angularity
	Perpendicularity
	Parallelism
	Position
	Concentricity
	Symmetry

	Circular Runout
	Total Runout

4. In the **Tolerance** section, select a symbol that you want to precede the tolerance value.

Use this symbol	To represent this
	Diameter
	Spherical Diameter

5. In the **Tolerance** dialog box, type a value for the tolerance.

Note:
You can use numeric and ASCII values.

6. In the **Tolerance** section, use the drop-down list box to select a modifier that applies to the tolerance.

Use this symbol	To represent this
	MMC — Maximum Material Condition
	LMC — Least Material Condition
	Free State
	Projected Tolerance Zone
	Tangent Plane
	Statistical Tolerance

You can select several modifiers. Subsequent modifiers are based on your initial modifier selection.

7. (If you have used the **Projected Tolerance Zone** modifier) Type a value in the box that appears below the **Projected Tolerance Zone** modifier.
8. In the **Primary Datum** section:

- Select a datum (**A-D**) from the list.
- Type one of these datum options:

Use this symbol	To represent this
	MMC - Maximum Material Condition
	LMC - Least Material Condition

9. (Optional, to create a compound datum) Choose the second **Primary Datum** menu, and select a secondary datum.

Note:

You can create up to six compound datums. After you select a primary datum, menus for a secondary datum appear. If a secondary datum is specified, menus for a tertiary datum appear. The secondary and tertiary datums can be compounded up to six times.

10. In the **KPC Symbol** section, type a number that appears inside the KPC symbol.

The KPC symbol displays to the right of the **Feature Control Frame** dialog box.

11. (Optional) To define a composite feature control frame, populate the composite tolerance input fields and datum fields to completely define it.

Note:

The characteristic field is limited to a positional or profile tolerance only.

12. (Optional) To clear symbols currently entered in the **GD&T Annotation Editor** dialog box, click **Clear All**.
13. To manage feature control frame setting, on the Feature Control Frame page of the **GD&T Annotation Editor** dialog box, select one of the following options:

Click this	To do this
Add to Favorites	Add the current settings to the Favorites combo box (up to 5).
Update Favorite	Change the setting of the selected favorite (shown in the combo box).
Clear All	Clear your current settings.

14. To insert the markup, click **OK**.

15. In the Viewing window, depending on your insertion mode, select a part to anchor to, a GD&T markup stack to add to, or any point in the view.

Create GD&T notes

Create notes that include GD&T symbols.

1. Do one of the following:

To	Do this
Anchor the markup to a part.	On the GD&T Markup toolbar , click Anchor Mode  .
To add the markup to a GD&T markup stack.	On the GD&T Markup toolbar , click Stack Mode  .
To place the markup anywhere in the view.	On the GD&T Markup toolbar , ensure Anchor Mode and Stack Mode are not active.

2. On the **GD&T Markup toolbar**, click **GD&T Annotation Editor** .
3. Navigate to the Notes page and in the **Text** section, enter your text and choose a symbol from the **Symbols** section (optional).

Use this symbol	To represent this
	Countersink
	Counterbore/Spotface
	Depth/Deep
	Conical Taper
	Slope
	Square
	Between
	Plus/Minus
	Degrees
	Diameter

	Spherical Diameter
	Statistical Tolerance

- 4. (Optional) To set the color of the characters and symbols within a note, click **Text Color** and select a color.
- 5. (Optional) Click **Clear All** to clear the text and symbol entries.
- 6. To insert the markup, click **OK**.
- 7. In the Viewing window, depending on your insertion mode, select a part to anchor to, a GD&T markup stack to add to, or any point in the view.

Create GD&T datums

Add GD&T datum feature and target symbols as 3D markups.

- 1. Do one of the following:

To	Do this
Anchor the markup to a part.	On the GD&T Markup toolbar , click Anchor Mode  .
To add the markup to a GD&T markup stack.	On the GD&T Markup toolbar , click Stack Mode  .
To place the markup anywhere in the view.	On the GD&T Markup toolbar , ensure Anchor Mode and Stack Mode are not active.

- 2. On the **GD&T Markup toolbar**, click **GD&T Annotation Editor** .
- 3. In the Datums page, in the **Datum Type** section, choose **Feature** or **Target**.
- 4. For **Feature**, in the **Datum Identifier** section, type the datum identifier letter that you want to use.

Note:
You can create double letter datums such as AA.

- 5. For **Target**, do the following:
 - a. In the **Size** section, type a value.

Use both numeric and ASCII characters.

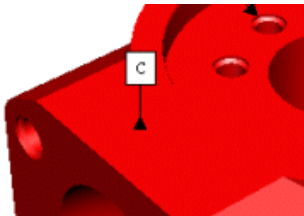
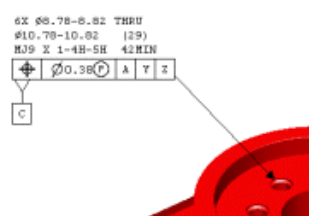
- b. In the **Identifier** section, type the identifier and, optionally, select a symbol from the following list:

Use this symbol	To represent this
∅	Diameter
□	Box

- (Optional) Use **Clear All** to clear text that you have entered.
- To insert the markup, click **OK**.
- In the Viewing window, depending on your insertion mode, select a part to anchor to, a GD&T markup stack to add to, or any point in the view.

Note:

3D datum features exhibit special behaviors. When creating a datum feature with **Anchor Mode** enabled, the base is represented by a backwards arrowhead on the leader line. When creating a datum feature with **Anchor Mode** disabled, the datum feature is displayed at a fixed location on the screen and will not move as the part is rotated. When stacking a datum feature onto an existing GD&T markup, the base is rendered as part of the flat-to-screen markup, and aligned to the left of the markup above it in the stack.

		
A datum feature with Anchor Mode enabled	A datum feature with Anchor Mode disabled	A datum feature stacked upon an existing markup

Create GD&T tolerances

- Do one of the following:

To	Do this
Anchor the markup to a part.	On the GD&T Markup toolbar , click Anchor Mode  .
To add the markup to a GD&T markup stack.	On the GD&T Markup toolbar , click Stack Mode  .
To place the markup anywhere in the view.	On the GD&T Markup toolbar , ensure Anchor Mode and Stack Mode are not active.

2. On the **GD&T Markup toolbar**, click **GD&T Annotation Editor** .
3. In the Tolerance page, choose one of the following formats from the **Tolerance** section:

- Equal Bilateral Tolerance 
- Unequal Bilateral Tolerance 
- Tolerance Limits 

Input fields in the Tolerance page are selected or cleared depending on the format you selected.

4. In the **Instances** section, click **X** to activate the text box, and type the number of tolerance instances this tolerance applies to.
5. In the **Leading Note** section, include both **Symbols** and text for your note.
6. (Optional) In the **Feature Size** section, click the diameter symbol and enter a value in the text box.
7. (Optional) In the **Upper Limit** section, type a value.
8. (Optional) In the **Lower Limit** section, type a value.
9. In the **Trailing Note** section, include both **Symbols** and text of your note.
10. (Optional) Use **Clear All** to clear all text.
11. To insert the markup, click **OK**.
12. In the Viewing window, depending on your insertion mode, select a part to anchor to, a GD&T markup stack to add to, or any point in the view.

Create a GD&T markup stack

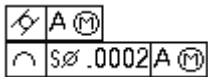
1. If necessary, create the first GD&T markup of the stack.
2. On the **GD&T Markup** toolbar, click **Stack Mode** .
3. On the **GD&T Markup** toolbar, click **GD&T Annotation Editor**, and create the next markup of the stack.
4. Click **OK**.

The cursor changes icons, indicating the specific GD&T markup is ready to be stacked on another markup in the Viewing window.



5. In the Viewing window, click the target markup or markup stack.

The new GD&T markup symbol is stacked below the selected GD&T markup or markup stack. Both symbols will move together when dragged.



6. (Once the markups are stacked) In the Viewing window, right-click a GD&T markup, and select one of the following options:

Use this option	To do this
Bring to Front	Move the selected markup to the front of the other markups.
Send to Back	Move the selected markup behind the other markups.
Move Up	Move the selected markup higher on the stack.
Move Down	Move the selected markup lower on the stack.
Unstack	Remove the markup from the stack.
Edit	Bring up the GD&T Annotation Editor dialog box.

Use this option	To do this
Delete	Delete the markup.
Properties	Open the GD&T Markup Properties dialog box.

Note:

The gap between a datum feature symbol that is stacked to another feature symbol is used to draw the backwards arrowhead base. Use this area to attach the other markup symbol.

Edit GD&T markups

1. In the Viewing window, right-click a GD&T markup, and choose **Edit**.

The markup appears in the **GD&T Annotation Editor** dialog box.

2. Edit the markup and click **OK**.

Display and hide 3D GD&T markups

You can display, hide, and rearrange 3D GD&T markups.

Do one or more of the following:

To	Do this
Display or hide individual 3D layers or 3D GD&T markups.	In the Assembly view, in the Markup Layers list, select or clear the check box to the left of the markup layer or markup name  .
Rearrange 3D GD&T markups.	In the Assembly view, in the Markup Layers list, click the markup you want to move and drag it to its new position.

Change GD&T markup preferences and properties

You can change appearance preferences that apply to all new GD&T markups. You can also modify the appearance properties of an existing GD&T markup.

1. On the **GD&T Markup** toolbar, click **Preferences** .
2. In the **GD&T Markup Default Preferences** dialog box, set the preferences for edge, fill, or font.
3. (Optional) To modify the appearance of an existing GD&T markup, right-click the markup and select **Properties**.

Use the **GD&T Markup Properties** dialog box to modify existing markup properties, for example **Edge**, **Fill**, and **Font**.

Save 3D GD&T markups

3D GD&T markups are managed by the **3D markup layer system**. GD&T markup types are listed as **Notes**, **Datum Features**, **Datum Targets**, and **Tolerances**.

The node of each symbol controls the visibility of each markup in the tree. When GD&T symbols are stacked together, changing the visibility of a single entity in the stack changes the visibility of the entire stack. The stack can only be either all on or all off.

To save the GD&T markup, you must **save the 3D markup layer**. Each markup layer is saved as a separate file.

Associate 3D GD&T markups to a view

You can associate a view orientation to an individual markup layer. You can then create a series of markup layers, each with its own specific camera orientation. Doing so simulates a multi-view drawing using the 3D geometry.

1. To assign a view orientation, in the Viewing window, right-click the markup layer, and select **Capture View**

The view saves the camera orientation and visible parts. A 3D frame outlining the view and a label with the view name are created.

2. To orient the model to the assigned view, in the Viewing window, right-click the markup layer, and select **Apply View**.

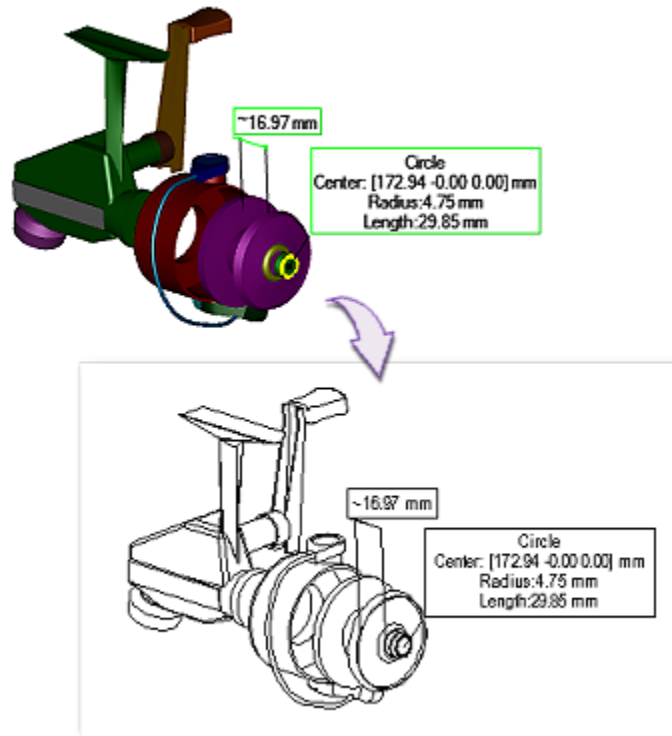
The model is oriented to the captured view.

3. To close the 3D frame outline and label, in the Viewing window, right-click the markup layer, and select **Hide View**.

9. Capturing outline drawings of models

Overview of capturing outline drawings of models

Use outline capture to capture outline drawings of your model. The outline drawing accurately reflects the current 3D model view, and includes any measurements, markups, constraint annotations, and the center line that are visible in the view. Each of these elements is captured on different layers.



Set outline capture preferences to control the appearance of the outline drawing.

Tip:

When you have captured an outline drawing, use the 2D tools to view and modify the resulting 2D image.

Create an outline

You can create a line drawing that accurately represents your 3D model. An outline capture can consist of parts, measurements, and markups.

1. Display the 3D model as you want it to appear in the line drawing.

2. (Optional) Change the outline capture preferences to control the appearance of the line drawing.
3. Choose **Actions→Outline Capture**, and then click one of the following options:
 - To create a line drawing of the model in the active Viewing window only, choose **Current 3D Window**.
 - To create line drawings of the open 3D models, choose **All 3D Windows**.

A 2D image appears, with a background image and an outline layer.

Overview of setting outline preferences

Set the outline preferences to control the appearance of lines in an outline drawing. Use part highlighting to emphasize parts by specifying the shape, color, and thickness of their lines.

You can create halos around text boxes, circles, and leader lines that connect markups. A halo is a surrounding area where nothing is drawn.

Note:

A halo around a leader line that crosses another line will cause the line representing the part to break at the intersection point, helping to distinguish which parts are connected to the markup.

Set outline preferences

1. Choose **Actions→Outline Capture→Preferences**.
2. In the **3D Geometry Asset Preferences** dialog box, on the **Outline** tab, click **Outline Capture Preferences**.
3. In the **Outline Preferences** dialog box, choose the appropriate preferences.

Note:

To have units other than pixels, you need to enable **page extents**.

On this page	Use this option	To do this
General	Feature Angle	Specify the degree of the angle necessary before a line is drawn between two adjacent surfaces. If the angle between the normals of two adjacent surfaces equals or exceeds the Feature Angle value, the two surfaces are considered separate features and the edge between them is drawn. If the angle between the normals of two adjacent

On this page	Use this option	To do this
		surfaces is less than the Feature Angle value, the two surfaces are considered one feature and the edge between them is not drawn. To draw more edges, decrease the value in Feature Angle. To draw fewer edges, increase the value in the Feature Angle. Use integers 0-180.
	Line Thickness	Specify the thickness of the outline. To draw a thicker line, increase the Line Thickness value; to draw a thinner line, decrease the Line Thickness value.
	Include lines for rounded edges	Draw edges that are rounded. Clear Include lines for rounded edges to draw only silhouette lines, horizon lines, and feature lines as defined by the Feature Angle value. Select Use all B-Rep edges to display all the B-Rep edges in the outline image.
	Remove hidden lines	Draw only the edges that are visible in the front of the model in the outline image. Clear Remove hidden lines to draw the edges that are hidden behind or in the back of the model as well.
	Preserve colors	Display the outline in the same colors as the model parts, or clear Preserve colors to draw all the parts with one color. If you clear Preserve colors, the colors of the background and outlines in the 2D image are determined by the default 2D background color and line properties.
	Override background color	Replaces the current background color with the color of your choice in the outline image.
	Create outline using Parasolid B-Rep if available	If a part in the model contains Parasolid B-Rep (XT B-Rep data), then you can use this option to create an outline. The outline will be more precise because it is based on the CAD data and not on the tessellated version of the part.
Advanced	Line Style	Specify the shape of the lines that are drawn to represent highlighted parts.
	Line Color	Specify the color of the lines that are drawn to represent highlighted parts.
	Thickness	Specify the thickness of the lines that are drawn to represent highlighted parts. Use integers between 0 and 999.

On this page	Use this option	To do this
	Disable Highlighting for All Parts	Draw selected parts as if they are not selected. You can also disable highlighting for individual parts.
	Enable Hotspot Creation (CGM Only)	Activate hotspots for anchored text markups.
	Leader Lines Just Touch Silhouette Edges	Draw the leader lines of anchored text markups so that they just touch the edges of the lines representing parts.
	Add Halos to Leader Lines	When leader lines cross lines that represent parts, draw a halo around the leader line so that the line representing the part appears to break at the intersection point.
	Halo Thickness	Specify the thickness of the halo. Use integers between 1 and 999.

- Click **OK**.

New outlines reflect the preferences you have set. Existing outlines do not change.

Enable outline highlighting for a single part

In the Viewing window, right-click the part and choose **Enable Outline Highlighting**.

Override the default highlighting preferences for a single part

- In the Viewing window, right-click the part, and then click **Override Outline Highlighting**.
- In the **Highlighting Preferences** dialog box, choose the appropriate preferences.

Use this option	To do this
Override Default Highlighting Preferences	Change the line style, color, or thickness from the values you specified on the Advanced page of the Outline Preferences dialog box.
Line Style	Specify the shape of the lines that are drawn to represent highlighted parts.

Use this option	To do this
Line Color	Specify the color of the lines that are drawn to represent highlighted parts.
Thickness	Specify the thickness of the lines that are drawn to represent highlighted parts. Use integers between 0 and 999.

3. Click **OK**.

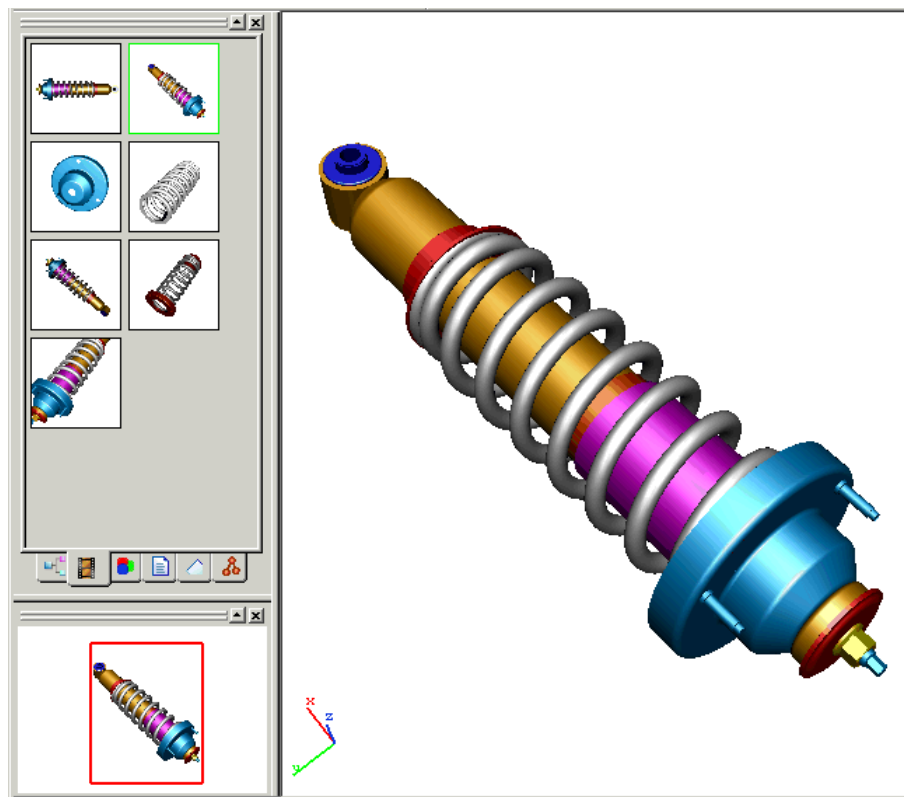
The lines that represent the part appear as you have specified.

10. Working with 3D snapshots

Overview of 3D snapshots

Snapshots capture the state of the 3D Viewing window. If you capture snapshots of several views of your model, you can return to any view by selecting its snapshot. You can apply captured 3D elements selectively or all at once. When you work with a sequence of snapshots, you can establish relationships between snapshots and create an animation by cycling through the series.

Snapshots appear in the **Snapshots** view. You can display snapshots as thumbnail images or as a tree. All tasks that you can perform while in thumbnail mode you also can do in tree view mode; however, only tree view mode displays each of the 3D elements that are present in your snapshots.



What snapshots can capture

Many elements contribute to your view of a model. You can capture the current states of any or all of the following:

Use this option	To capture
Active Hierarchy	Original assembly hierarchy or an alternate hierarchy.
CAE	CAE results and annotations.

Use this option	To capture
Camera Orientation	Camera point, target point, and orthographic or perspective projection mode.
Compare	Compare recipes.
Constraint	3D constraints.
Coordinate System	All user defined coordinate systems.
Cross Sections	All cross section planes.
Exploded Views	All named explosions.
Hide Obscuring	Views created by hiding obscuring parts.
Jack	Jack information.
Layer Filters	The active layer filter.
Light Assignments	All light sources and the current state of each light.
Markup layers	Visible and invisible markup layers.
Material Assignments	Part color, shininess, transparency, and advanced appearance properties.
Measurements	Visible measurement labels. Note: By default, "Restored Measurement" is added to labels of non-associative measurements and associative measurements that have not been updated to indicate that the measurement may be out-of-sync with the 3D data.
Part Display Mode	Display modes set for individual parts. The current default display mode is not captured.
PMI	Product and Manufacturing Information.
Reference Sets	The active reference set state.
Selection	Part selection state, or which parts and assemblies are selected. This does not include which part <i>features</i> are selected.
Texture Coordinate Assignments	Part appearance affected by 2D images mapped to 3D geometry.
Transforms	Current location, rotation, and scale of each part.
True Shading	True shading settings.
Visibility	Visibility of each part.
Visual Report	Visual reports.
Volume Clipping	Locations and properties of clipping volumes.

Note:

Your list of elements may differ from this list because some elements require advanced service levels or optional modules.

Capture and manipulate 3D snapshots

Use snapshots to capture the state of the 3D Viewing window.

1. (Optional) Specify snapshot preferences.
2. Choose **Window**→**Show View**→**Other**→**Visualization**→**Snapshots**.
3. Choose **Tools**→**Snapshots**.
4. Do any of the following:

To	Do this
Capture a snapshot	On the Snapshots toolbar, click Add  . -or- In the Snapshots view, right-click and choose Add .
Cycle through a series of snapshots	In the Snapshots view, right-click and choose Cycle .
Insert a snapshot into a series of snapshots	On the Snapshots toolbar, click Insert  . -or- In the Snapshots view, right-click and choose Insert .
Delete a snapshot	- In the Snapshots view, select the snapshot to delete. - On the Snapshots toolbar, click Delete . -or- Right-click the snapshot and choose Delete.

To	Do this
	Note: You can use the Undo command to undo the last 3D snapshot Delete action.
Replace a snapshot with a new snapshot Note: Using Replace is similar to deleting and recapturing a snapshot. The 3D elements captured in the snapshot reflect the current Snapshot Capture Preferences. If the snapshot being replaced contains 3D elements that are not currently enabled in the preferences, they are not included in the new snapshot. Using Replace also recaptures the thumbnail image displayed in the Snapshots view. The name of the snapshot is retained.	a. In the Snapshots view, select the snapshot to replace. b. On the Snapshots toolbar, click Replace . -or- Right-click the snapshot and choose Replace.
Update a snapshot Note: Using Update recaptures the 3D elements that are present in the snapshot. For 3D content to be updated, the appropriate preferences must be active in the Snapshot Capture Preferences. Only pre-existing 3D elements within the snapshot are modified. 3D elements not enabled in the preferences when the snapshot was initially generated are not created. Using Update also recaptures the thumbnail image displayed in the Snapshots view.	In the Snapshots view, right-click the snapshot and choose Update.
Recapture the snapshot image displayed in the Snapshots view.	In the Snapshots view, right-click the snapshot and choose Recapture Image.

To	Do this
Rename a snapshot	In the Snapshots view, right-click the snapshot and choose Rename .
Disable a snapshot from playing with a cycle of snapshots	In the Snapshots view, right-click the snapshot and choose Disable .
Assign references to a snapshot	a. In the Snapshots view, right-click the snapshot with which you want to associate other snapshots, and choose References. b. In the Add References dialog box, select the snapshots to reference. Cycling through a sequence of snapshots, a snapshot with references triggers its referenced snapshots.
In the Snapshots view, display snapshots in thumbnail mode or tree view mode.	On the Snapshots toolbar, click Toggle View  -or- In the Snapshots view, right-click and choose Toggle View.
Create a new window containing snapshots displayed in thumbnail mode	In the Snapshots view, right-click and choose Send To Window .
Change the thumbnail size.	In the Snapshots view, right-click and choose Thumbnail Size ; then choose Small , Medium , or Large .
Specify snapshot preferences	On the Snapshots toolbar, click Preferences  -or- In the Snapshots view, right-click and choose Preferences.

Apply snapshots

You can apply a snapshot's captured 3D elements selectively or all at once to the active Viewing window.

Tip:

Display the **Snapshots** view in the tree view mode to view a list of the 3D elements captured for each snapshot.

To	Do this
Apply snapshot elements all at once	Do one of the following: • (When the contents of the Snapshots view are displayed as images) Double-click a snapshot. • Right-click a snapshot and choose Apply→All.
Apply snapshot elements selectively	Right-click a snapshot, point to Apply, and choose one of the following: • Active Hierarchy • Behaviors • CAE • Camera Orientation • Compare • Constraint • Coordinate System • Cross Sections • Exploded Views • Hide Obscuring • Jack • Layer Filters • Light Assignments • Markup Layers • Material Assignments

To	Do this
	• Measurements • Part Display Mode • PMI • Reference Sets • Selection • Texture Coordinate Assignments • Transforms • True Shading • Visibility • Visual Report • Volume Clipping
Update a snapshot with elements from another snapshot	1. Double-click a snapshot. 2. Right-click another snapshot, point to Apply, and choose the captured elements that you want to add. 3. Right-click the first snapshot, and choose Update. The first snapshot is updated with the elements that you applied from the second snapshot.

Tip:

After applying a snapshot, you can add elements from another snapshot to the Viewing window. However, only the Viewing window is updated, not the initial snapshot.

Merge snapshots

You can merge elements from individual snapshots.

Note:

To merge snapshot elements, each snapshot must consist of mutually exclusive content. If two snapshots have an identical snapshot element and you apply one of the snapshots followed by the other, the contents of the first snapshot are replaced by the contents of the second snapshot. If two applied snapshots consist of unique snapshot elements, the effects of each snapshot are active simultaneously.

1. Set the Snapshot Capture Preferences to capture one or more snapshot elements present in the Viewing window.
2. On the **Snapshots** toolbar, click **Add** .

A snapshot is created which consists only of the specified elements.

3. Change the Snapshot Capture Preferences to capture snapshot elements that are not contained in your other snapshot.
4. On the **Snapshots** toolbar, click **Add** .

A second snapshot is created which consists of elements different from those contained within your first snapshot.

5. On the **Snapshots** view, double-click one of the snapshots.

The effects of the snapshot are added to the Viewing window.

6. Double-click the second snapshot.

The effects of the second snapshot are added to the Viewing window. If the snapshots consist of mutually exclusive content, all of the elements from both snapshots are present.

Note:

You also can merge snapshot elements selectively. Do the following to apply a snapshot element present in an inactive snapshot:

- a. On the **Snapshots** view, right-click and select **Toggle View**.
- b. Right-click a snapshot element and choose **Apply**.

Tip:

After merging snapshots, to create a new snapshot which consists of all of the active snapshot elements, update the Snapshot Capture Preferences to reflect the current state of the Viewing window, and then capture a new snapshot.

Scope snapshots to alternate hierarchies

You can limit the scope of a snapshot to the active **alternate hierarchy**.

1. Create an alternate hierarchy which consists of only the product structure that you want to be affected by the snapshot.
2. On the **Snapshots** toolbar, click **Preferences** .
3. In the **Snapshot Capture Preferences** dialog box, select the check box for each 3D element that you want to include in the snapshot.
4. Select the **Scope to Active Alternate Hierarchy** check box.
5. Click **OK**.
6. In the Assembly, ensure that only the alternate hierarchy is loaded and visible.
7. Create the snapshot.

Create snapshot references

You can create a link from one snapshot to other snapshots that contain unique or scoped content.

1. Do one of the following:
 - Create snapshots which consist of mutually exclusive content. Set the snapshot capture preferences so that each snapshot contains only unique snapshot elements.
 - Create snapshots scoped to different alternate hierarchies.
2. Create the snapshot that you want to function as a composite of your other snapshots.
3. Right-click the composite snapshot and choose **References**.
4. In the **Add References** dialog box, select the check box for each snapshot that you want to reference.

- Click **OK**.

The snapshot references are created. When you double-click your composite snapshot, the referenced snapshot elements are applied to the Viewing window.

Cycle through snapshots

You can cycle through a sequence of snapshots, starting with any snapshot in the sequence. A snapshot cycle begins with the selected snapshot, and ends with the last snapshot in the series.

Tip:

Click and drag a snapshot to another position in the sequence to change the playback order, or selectively include only those snapshots that you want to play, in the order that you want to play them. You also can disable any snapshots in the sequence.

To	Do this
Cycle through a snapshot sequence	- In the Snapshots view, select the snapshot with which you want to begin your sequence. - Right-click the selected snapshot, and choose Cycle.
Change the snapshot order	Select a snapshot, and drag it to another position.
Selectively include and order snapshots in a sequence	Hold Ctrl and use your mouse to click the snapshots that you want to include, in the order that you want to play them.
Disable a snapshot	Right-click a snapshot, and choose Disabled .

Select visible product structure

You can select all of the product structure that is visible in a snapshot. This enables you to use snapshots to highlight different sets of product structure in the Viewing window and in the **Assembly** view.

In the **Assembly** view, right-click a snapshot and choose **Select→Visibility**.

The product structure that is visible in the snapshot is selected.

Work with textures in snapshots

When a snapshot includes texture coordinates, it does not contain a copy of the texture coordinates. Instead, the snapshot contains an assignment to a set of texture coordinates that are defined in the appearance system. When you update a snapshot with changes to a model, such as transforming a model by scaling parts, a new set of texture coordinates is created for the snapshot. If you update

a second snapshot, another set of texture coordinates is created. Then if you want to change the parameters of the texture coordinates, you must do this for each snapshot.

To avoid updating each snapshot, you can define a single set of texture coordinates to use for a series of snapshots. Then, if needed, you can update specific parameters of those texture coordinates, which will take effect automatically in all of the snapshots.

Before you capture a snapshot with textures, you should determine whether or not the texture coordinates may change at some point. You define the texture coordinates for a snapshot differently based on whether the texture coordinates remain static or change.

Set up static texture coordinates

When you don't expect to make changes to the texture coordinates of your snapshots, you can turn off texture coordinate assignments for snapshots. The appearance system will automatically manage the texture coordinates contained in the models as you perform apply, scale, and update operations to the snapshots.

1. In the **Snapshots** view, right-click and choose **Preferences**.
2. In the **Snapshot Capture Preferences** dialog box, clear **Texture Coordinate Assignments**.
3. Click **OK**.

Set up texture coordinates that may change

When you expect to make changes to the texture coordinates of your snapshots, follow these steps to ensure consistency of texture coordinates used in snapshots.

1. Define a reference snapshot that contains only the texture coordinate assignments.
 - a. In the **Snapshots** view, right-click and choose **Preferences**.
 - b. In the **Snapshot Capture Preferences** dialog box, clear all options except **Texture Coordinate Assignments**, and then click **OK**.
 - c. In the **Snapshots** view, right-click and choose **Add** to add a new snapshot.
 - d. In the **Snapshots** view, right-click and choose **Toggle View**.

The snapshot appears in tree view mode.

- e. Expand the snapshot.

Only **Texture Coordinate Assignments** is listed.

- f. Right-click the snapshot, and choose **Rename**. Rename the snapshot to indicate that it is a reference snapshot for texture coordinate assignments.
2. Select the items to be captured in the snapshots.
 - a. In the **Snapshots** view, right-click and choose **Preferences**.
 - b. In the **Snapshot Capture Preferences** dialog box, select the options to include in the snapshot captures. Make sure **Texture Coordinate Assignments** is not selected. Then click **OK**.
 - c. Set up the view as desired for a snapshot, and in the **Snapshots** view, right-click and choose **Add**.
3. Add the texture coordinate assignments reference snapshot to snapshots.
 - a. Right-click the new snapshot, and choose **References**.
 - b. In the **Add References** dialog box, select the texture coordinates snapshot, and then click **OK**.

In the tree view mode, the **References** item appears under the snapshot, and it contains the texture coordinate assignments snapshot.

- c. Add the texture coordinate assignments snapshot to other snapshots as desired.

Note:

If you have existing snapshots with texture coordinate assignments, you can replace the assignments with the texture coordinate assignments snapshot you just created:

- A. Expand the existing snapshot.
- B. Right-click the **Texture Coordinate Assignments** item, and choose **Delete**.
- C. Right-click the pre-existing snapshot and choose **References**.
- D. In the **Add References** dialog box, select the check box for the texture coordinate assignments snapshot.
- E. Click **OK**.

Update snapshots that use a reference snapshot for texture coordinate assignments

Once you have a reference snapshot set up for texture coordinate assignments, any updates to the model take effect automatically in all of the snapshots; however, the first time you make changes to a model that uses a reference snapshot, you must capture the reference snapshot.

1. **Apply** all or some snapshot elements to a snapshot.

You do not have to apply **Texture Coordinate Assignments**.

2. Update the model as desired, for example, transform a part.
3. In the **Snapshots** view, in tree view mode, expand the texture coordinate assignments snapshot.
4. Right-click the **Texture Coordinate Assignments** item, and choose **Capture** to update the reference snapshot.
5. Right-click the snapshot to which you applied elements in step 1, and choose **Update**.
6. Apply elements to other snapshots, and update the model as desired.

You do not have to capture the reference snapshot again. The correct texture coordinate assignments are applied for all subsequent updates.

Using PLM XML Product Views to manage part visibility

You can save snapshots as PLM XML Product Views to control part visibility in the 3D Viewing window (excluding Base).

Merge PLM XML Product Views with snapshots to control part visibility in the 3D Viewing window (Base).

1. Open a JT or PLM XML file that contains product structure that matches your Product Views.
2. Choose **File→Merge**.
3. In the **Merge File** dialog box, open the PLM XML file that contains your Product Views.

In the **Snapshots** view, the Product Views appear as snapshots.

Save snapshots as Product Views

1. Set the snapshot preferences to capture part visibility.
2. Capture snapshots of each product variation that you want to save.
3. Choose **File→Export**.
4. In the **Export** dialog box, from the **File format** list, select **Product Structure (*.plmxml)**.
5. Click **Options**.

- (Optional) On the **General** page of the **PLM XML export** dialog box, select the **Product Structure Data** check box to include the product structure of the model in the exported .plmxml file.

Note:

If you do not save the model product structure, the exported .plmxml file cannot be opened on its own. It must be merged with a compatible JT or PLM XML assembly.

- Select the **Product View Data** check box to include snapshots in the exported .plmxml file.
- (Optional) Select the **Current State of Document** check box to capture a Product View of the current state of the document. You can specify to include any of the supported snapshot elements.

Note:

A current state Product View is automatically applied when you load the .plmxml file. It does not appear in the **Snapshots** view.

- On the **Product View** page, select the check box for each snapshot that you want to save as a Product View.
- Click **OK**.
- In the **Export** dialog box, click **OK**.

A .plmxml file is created. Your snapshots are included in the file as Product Views. If you included the model product structure, you can open the file and access the Product Views in the **Snapshots** view.

Merge Product Views with a compatible JT or PLM XML assembly

- Open a JT or PLM XML file that contains product structure that matches your Product Views.
- In My Teamcenter, right-click the PLM XML file that contains your Product Views and choose **Send To→Lifecycle Viewer**.

In the **Snapshots** view, the Product Views appear as snapshots.

Save snapshots

If you capture snapshots and want them to be available the next time you work with the model, save your work to a session file or a .plmxml file.

Do one of the following:

To	Do this
Save snapshots to a session file	Note: Snapshots automatically save when you save to a session file. 1. Open a model and capture one or more snapshots. 2. Choose File→Save Session.
Save snapshots to a .plmxml file	Note: You can choose to include snapshots when you save your work to a .plmxml file. 1. Choose File→Preferences→PLM XML. 2. On the Save page of the PLM XML Preferences dialog box, select the Save extended 3D content into PLM XML check box, and then click OK. 3. (Optional) To save the .plmxml file without a default product view, clear the Save the default snapshot checkbox. 4. Choose File→Save As. 5. Type a file name and click OK.

Specify snapshot preferences

Specify which 3D elements to include in your snapshots.

1. In the **Snapshots** view, right-click and choose **Preferences**.
2. Choose any of the following:
 - **Active Hierarchy**
 - **Behaviors**
 - **CAE**
 - **Camera Orientation**
 - **Compare**

- **Constraint**
- **Coordinate System**
- **Cross Sections**
- **Exploded Views**
- **Hide Obscuring**
- **Jack**
- **Layer Filters**
- **Light Assignments**
- **Markup Layers**
- **Material Assignments**
- **Measurements**
- **Part Display Mode**
- **PMI**
- **Reference Sets**
- **Selection**
- **Texture Coordinate Assignments**
- **Transforms**
- **True Shading**
- **Visibility**
- **Visual Report**
- **Volume Clipping**

3. (Optional) Select the **Scope to Active Alternate Hierarchy** check box.

11. Working with motion

Managing motion files

Overview of motion files

Add movement to your 3D models with VFM and PLM XML motion files. You can animate your models to illustrate assembly sequences or to show them off during design reviews. You also can create motion files for use with **path planning** or **clearance analysis**.

To animate models, you must have the Professional or Mockup service level.

Begin animating your models with motion files by reviewing the following subjects:

- Creating motion files
- Playing motion files
- Changing part assignments
- Use the Paths page
- Optimize motion files
- Save motion files
- Converting .xmo and .res motion files
- .xmo motion file specifications
- Specify motion preferences

Creating motion files

Overview of creating motion files

When animating your 3D models, you can

- Create VFM or PLM XML motion files.
- Create motion paths to guide your parts through the 3D Viewing window.
- Work with key frames along motion paths.

- Position your geometry in the 3D Viewing window.
- Manipulate motion paths in the 3D Viewing window.
- Add steps between key frames.
- Copy motion paths from one motion file to another.
- Edit existing motion files.
- Modify a path assigned to inactive product structure.

Create motion files

Create a motion file to show part movement in the 3D Viewing window.

Note:

To create motion files, you must have the Professional or Mockup service level.

1. On the **3D Motion** toolbar, click **Preferences** .
2. In the **Motion Preferences** dialog box, make sure the **Use selected parts to define path origin and assignment** check box is selected, and then click **OK**.
3. **Select** the parts that you want to move.
4. On the **3D Motion** toolbar, click **New Motion File** .

-or-

In the **Paths** view, right-click **Motion Files** and choose **New Motion File**.

A new motion file appears in the **Paths** view, along with a new path for your selected parts.

5. In the **Paths** view, click the + sign to the left of the path to display the path origin.
6. Add a **key frame**.

In the 3D Viewing window, the origin and the key frame are displayed as trihedrons to show the orientation of the parts.



The path origin and the first key frame (parts hidden)

7. (Optional) Position your geometry for the first key frame.
8. (Optional) Interactively modify the path in the 3D Viewing window.
9. (Optional) Change the number of steps between key frames to make the motion playback between the origin and the first key frame more or less smooth.
10. Add more key frames until your path is complete.
11. (Optional) Create additional paths.
12. Preview the motion or add it to the playlist.
13. Save the motion file.

Create motion paths

Create motion paths for parts or assemblies that you want to animate in the 3D Viewing window.

1. Create or edit a motion file.
2. In the **Paths** view, select the motion file.
3. On the **3D Motion** toolbar, click **New Path** .

-or-

Right-click the motion file and choose **New Path**.

A new path appears.

Add, edit, or delete key frames

Use key frames to define positions for your parts along a motion path.

1. Create or edit a motion file.

2. In the **Paths** view, click the + sign to the left of the path to display the path origin along with any existing key frames.
3. Do any of the following:

To	Do this
Add a new key frame to the end of the path	On the 3D Motion toolbar, click Insert Key Frame . -or- In the Paths view, right-click the path and choose Insert Key Frame. The last existing key frame on the path is highlighted.
Insert a new key frame before or after an existing key frame	In the Paths view, right-click a key frame and choose Insert or Insert After.
Edit an existing key frame	In the Paths view, select the key frame, and then on the 3D Motion toolbar, click Edit Key Frame . -or- In the Paths view, right-click the key frame and choose Edit.
Delete a key frame	In the Paths view, select the key frame, and then click Delete  on the 3D Motion toolbar. -or- In the Paths view, right-click the key frame and choose Delete. Note: You can use the Undo command to undo the last Delete action.

4. (If adding or editing a key frame) In the **Key Frame** dialog box, change the position of the geometry you want to animate.

Position geometry in the Viewing window

When adding or editing the key frames of motion file paths, you must change the positions of the parts or assemblies you want to animate in the 3D Viewing window. When you **play** the motion, your geometry moves from key frame to key frame along the motion path.

Tip:

If your part movement in the Viewing window is too jerky, increase the number of **steps** between key frames.

1. Add or edit a key frame in an existing motion path.
2. In the **Key Frame** dialog box, do any of the following:

To position geometry	Do this
With a manipulator	a. Click Manipulator. A manipulator appears around the selected parts. b. Use the manipulator to translate or rotate the parts. Each time you use the manipulator to adjust your parts in the 3D Viewing window, a new key frame is added to the path. c. When you are finished creating key frames, click Manipulator Off.
With the Part Transformation dialog box	a. Click Transformation. b. Use the Part Transformation dialog box to translate the parts. When using the Part Transformation dialog box to adjust parts in the 3D Viewing window, a new key frame is added to the path when you click Apply. c. When you are finished creating key frames, click OK to close the Part Transformation dialog box, and then, in the Key Frame dialog box, click Transformation Off.
By picking locations on the model	a. Click Pick. b. Pick a location on the model. A new key frame is added each time you pick a different location. c. When you are finished creating key frames, click Cancel Pick.
By typing translation and rotation values in the Key Frame dialog box	a. If the Translation and Rotation sections are not visible, click Expand. b. (To change the position of your parts or assemblies in the Viewing window) In the Translation section, in the X, Y, and Z boxes, type translation distances. -or-

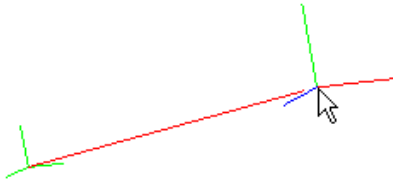
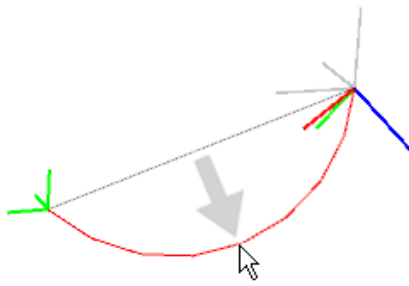
To position geometry	Do this
	Click Pick, and pick a location on your model. c. (To rotate your parts or assemblies) In the Rotation section, do any of the following: • (To change the angle of rotation) In the Angle box, type a value. -or- Click Pick, and pick a location on your model. • (To change the rotation axis) For Axis, in the X, Y, and Z boxes, type new coordinates. -or- Click Pick, and pick a location on your model. • (To change the center of rotation) For Center, in the X, Y, and Z boxes, type new coordinates. -or- Click Pick, and pick a location on your model. • (To pick a new axis and center) Click Pick Axis and Center, and pick a location on your model.

Modify paths interactively

You can edit paths in the 3D Viewing window.

1. Create a path with key frames.
2. In the **Paths** view, select the check box to the left of the path that you want to see in the Viewing window.
3. On the **3D Motion** toolbar, click **Preferences** .
4. In the **Path Interaction** section of the **Motion Preferences** dialog box, select the **Enable path manipulator** check box.

5. In the 3D Viewing window, do any of the following:

To	Do this
Select a key frame	Click the point on the key frame trihedron where the axis lines converge. The key frame and path segment are selected.  Select key frames
Move a key frame	Select a key frame, and drag it to a new position.  Move key frames
Change the rotation angle for a path segment	a. Select a key frame. b. Click and drag the path segment.  Change the rotation angle Note: This behavior is disabled if Show axis of rotation for selected key frame is turned on in the motion preferences.

To	Do this
	When the axis of rotation () is displayed, you can click and drag it to change the rotation angle.
Add a key frame	a. Right-click the point on the key frame trihedron where the axis lines converge and choose Insert or Insert After. b. Click and drag the new key frame to a new position.  Add key frames
Delete a key frame	Right-click the point on the key frame trihedron where the axis lines converge and choose Delete .
Reset a key frame to its previous state	Right-click the point on the key frame trihedron where the axis lines converge and choose Reset .
Lock a key frame	Right-click the point on the key frame trihedron where the axis lines converge and choose Lock .

Change the amount of key frame steps

Change the number of steps between key frames to adjust the smoothness of the motion playback. You can specify the default number of steps to create each time you add key frames, and you also can change the number of steps between any two key frames in sequence within a path. For smoother motion, increase the number of steps.

Note:

You can change the number of key frame steps only. You cannot change the location of steps, or create or delete them individually.

Tip:

If you want to see key frame steps in the 3D Viewing window while working on a path, you must specify to display them in the motion preferences.

Do either of the following:

To	Do this
Specify the default number of steps to create between key frames	1. On the 3D Motion toolbar, click Preferences . -or- - In the Paths view, right-click Motion Files and choose Preferences. 2. In the Motion Preferences dialog box, in the Default number of steps for key frames box, type how many intermediate steps you want created for each set of key frames.
Specify the number of steps to create between two specific key frames	1. In the Paths view, locate two frames in sequence for which you want to change the number of steps between them. 2. Right-click the second key frame and choose Edit. 3. In the Key Frame dialog box, in the Number of Steps box, type the number of steps you want separating the two adjacent key frames. Tip: If you want to change the number of steps between other sets of key frames, click Previous or Next to move through the positions on the path.

Copy motion paths

Reuse motion paths by copying them and then pasting them into motion files that reference similar product structure.

1. In the **Paths** view, right-click a path and choose **Copy**.
2. Right-click an appropriate motion file, and click **Paste**.

Edit motion files

You can update the paths and key frames of existing motion files.

Note:

You can edit VFM and PLM XML motion files. To work with XMO and RES files, you must first convert them to the VFM format.

1. Open the model associated with the motion file.

2. On the **3D Motion** toolbar, click **Open Motion File** .

-or-

In the **Paths** view, right-click **Motion Files** and choose **Open Motion File**.

3. In the **Select Motion Files** dialog box, open a VFM or PLM XML motion file.

The motion file appears in the **Paths** view.

4. Do any of the following:

To	Do this
Edit a path	a. In the Paths view, right-click the path and choose Edit. b. In the Key Frame dialog box, create or edit key frames.
Create a path	In the Paths view, select the motion file and then create the path.
Delete a path	In the Paths view, select the path, and then click Delete  on the 3D Motion toolbar. -or- In the Paths view, right-click the path and choose Delete. Note: You can use the Undo command to undo the last Delete action.

Modify a path assigned to inactive structure

Motion files associated with 3D models containing NX reference sets or model layers may have inactive target parts if the appropriate reference set or model layer is not active. This situation occurs when target parts are created with component bodies (sub-solids); if the model is opened using a reference set or layer filter where these bodies are inactive, the target parts are not loaded and the motion path cannot be edited.

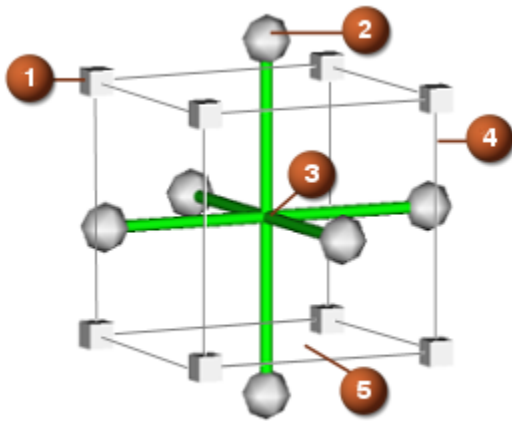
When authoring motion files, always associate target parts with geometry at the component level, rather than with component bodies. If this practice is followed, you should not encounter any issues when editing a motion path.

To resolve this problem, take one of the following actions:

- Change the **reference set load order**, and then close and reopen the file.

- Apply a different **layer filter**.
- Change the **product structure assignment**.

Manipulate parts in key frames



- 1 Click a square and drag to resize
- 2 Click a sphere and drag to rotate
- 3 Rotation point
- 4 Edge
- 5 Face

- Adjust manipulators in one or more of these ways:

To	Do this
Move the manipulator parallel to a face	Drag the face of the manipulator.
Move the manipulator perpendicular to a face	Press Ctrl as you drag the face.
Constrain the move to one axis of a face	Press Shift as you drag a face.
Resize the manipulator uniformly along all axes	Drag a corner of the manipulator.
Resize the manipulator along one axis	Press Ctrl as you drag a corner of the manipulator.
Rotate the manipulator around its X, Y, or Z axis	Drag an edge that is parallel to the axis.
Rotate the manipulator around the X, Y, or Z axis of the green jack	Drag a sphere handle in any direction.
Rotate the manipulator around the center of the green jack, in any direction	Press Ctrl as you drag a sphere handle in any direction.
Move the green jack (and its center of rotation)	Do one of the following: • Right-click a face, edge, or corner of the manipulator, choose Move Jack, and then click anywhere on the

To	Do this
	manipulator. The green motion jack aligns to the face, edge, or point. Press Shift as you click a sphere handle and drag the green motion jack to the new rotation point.
Align a point on the manipulator with a point on your model	- Right-click the face, edge, or corner of the manipulator that you want to align, and then choose Set Position. - Click a point on your model.
Rotate the manipulator to be parallel to a plane defined by two edges of your model	- Right-click the face, edge, or corner of the manipulator that you want to align, and then choose Set Orientation. - Click two edges.
Move and rotate the manipulator to align it to a point on your model and a plane defined by two edges of your model	- Right-click the face, edge, or corner of the clipping manipulator that you want to align, and then choose Set Composite. - Click a point on your model. - Click two edges.

Playing motion files

Overview of playing motion files

When you load motion files, they are initially displayed only in the **Paths** view. To play motion files, you must add them to the play list. Motion files on the play list can be played back in sequence or simultaneously.

Playing motion files involves the following tasks:

- Open motion files
- Preview motion files
- Create a play list
- Play motion files
- Return parts to their original positions

Open motion files

Open your motion files to play them in the 3D Viewing window or to edit them.

1. Open the model associated with the motion file.
2. On the **3D Motion** toolbar, click **Open Motion File** .

-or-

In the **Paths** view, right-click **Motion Files** and choose **Open Motion File**.

3. In the **Select Motion Files** dialog box, open a .vfm or .plmxml motion file.

The motion file appears in the **Paths** view.

Preview motion files

You can quickly preview a motion file without adding it to the play list. For motion files consisting of multiple paths, you can preview paths individually or simultaneously.

1. Open a motion file.
2. In the **Paths** view, select the motion file or a path.
3. On the **3D Motion** toolbar, click **Preview** .

-or-

In the **Paths** view, right-click the motion file or path and choose **Preview**.

The **Motion Preview** controls appear.

4. Play the motion file or path.

Create a play list

Create a play list to view your motion files in the 3D Viewing window. You can play multiple motion files in sequence or simultaneously.

Do any of the following:

To	Do this
Create a play list using the Motion Playback dialog box	1. On the 3D Motion toolbar, click Motion Playback . 2. In the Paths view, select a motion file. 3. In the Motion Playback dialog box, click Add.
Create a play list using the Paths view	In the Paths view, right-click a motion file and choose Add to Playlist. -or- (For multiple motion files) Right-click Motion Files and choose Add to Playlist to add all open motion files to the play list.
Automatically add motion files to the play list when you open them	1. On the 3D Motion toolbar, click Preferences . -or- In the Paths view, right-click Motion Files and choose Preferences. 2. In the Motion Preferences dialog box, select the Add to the play list when loaded check box.
Play motion files simultaneously	In the Motion Playback dialog box, from the Mode list, choose Parallel .
Play motion files in sequence	1. In the Motion Playback dialog box, from the Mode list, choose Sequence. 2. In the play list, select a motion file and click Move Up or Move Down to change the play list order. 3. Right-click a motion file and choose Edit to add a delay between the playback of motion files. In the Edit Playback Information dialog box, type the number of seconds you want to elapse before the file plays, and click OK.
Delay motion file playback.	1. In the Motion Playback dialog box, in the Play List section, double-click the motion file you want to delay. 2. In the Edit Playback Information dialog box, in the Delay box, type the number of frames you want to pass before the motion file begins to play. 3. Click OK.
Remove a motion file from the play list	In the play list, select a motion file, and then click Remove .

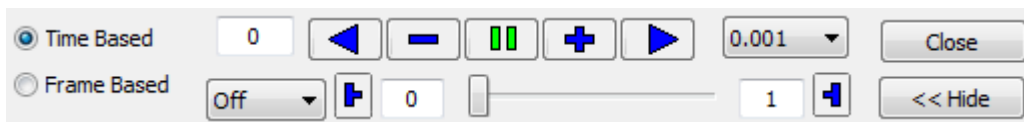
Play motion files

You can play motion files forward or backward, jump to a specific frame, fast forward, rewind, and repeat the animation.

1. Open or create a motion file.
2. Create a play list.
3. On the **3D Motion** toolbar, click **Motion Playback** .
4. In the **Motion Playback** dialog box, select the option for playback type:

- **Time Based**

The motion is played back based on the length of the path.



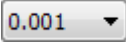
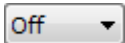
- **Frame Based**

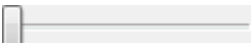
The motion is played back by displaying a frame for each point on the path.



5. Do any of the following to play the motion file:

To	Do this
Move to a specific frame or position.	In the Current Frame/Position 0 box, type the frame number (for frame based mode) or position (for time based mode).
(Frame based mode only) Play the motion backward, including even-numbered frames only.	a. Click Fast Reverse . b. (Optional) In the Start Frame and End Frame boxes, type the frame numbers with which you want to begin and end the playback.

	Tip: Each time you click Fast Reverse  or Fast Forward , the playback rate is doubled.
Play the motion backward from the current frame/position.	a. Click Reverse . b. (Optional) In the Start Frame/Position and End Frame/Position boxes, type the frame numbers/positions with which you want to begin and end the playback.
Move backward one frame/position from the current one.	Click Back One Frame/Position  .
Stop the motion playback.	Click Stop  .
Move forward one frame/position from the current one.	Click Forward One Frame/Position  .
Play forward from the current frame/position.	a. Click Play . b. (Optional) In the Start Frame/Position and End Frame/Position boxes, type the frame numbers/positions with which you want to begin and end the playback.
(Frame based mode only) Play forward, including even-numbered frames only.	a. Click Fast Forward . b. (Optional) In the Start Frame and End Frame boxes, type the frame numbers with which you want to begin and end the playback. Tip: Each time you click Fast Reverse  or Fast Forward , the playback rate is doubled.
(Time based mode only) Adjust the speed of the playback.	From the Adjust Speed  list, select the value to change the rate of the playback. x0.1 and x0.5 slow the playback by those values. The other options speed up the playback by those values.
Repeat the motion playback.	a. From the Loops  list, select Restart or Swing. b. Click Play or Reverse.

Start playback at a specific frame/position.	a. In the Start Frame/Position 0 box, type the frame number/position with which you want to begin playback. b. Click Play or Reverse .
Stop playback at a specific frame/position.	a. In the End Frame 6 box, type the frame number/position with which you want to stop playback. b. Click Play or Reverse .
Move to the first frame/position.	Click Back To First Frame/Position  .
Move to the last frame/position.	Click Forward To Last Frame/Position  .
Use the slider bar to control motion playback.	Move the Motion Slider Bar  .
Display the play list.	Click Expand .
Hide the play list.	Click Hide .
Close the Motion Playback dialog box.	Click Close .

Return parts to their original positions

After playing a motion file, reset the model parts to their positions before the motion playback.

On the **3D Motion** toolbar, click **Reset All Parts** .

Changing part assignments

Overview of changing part assignments

If motion files fail to resolve to your loaded product structure, you must update the motion assignment before you can view the animation.

Identify part assignments

Select parts associated with motion to identify them in the Viewing window. You can identify all of the parts associated with a motion file or you can identify parts associated with individual paths.

In the **Paths** view, right-click a motion file or a path and choose **Select Parts**.

In the Viewing window, the parts are selected.

Tip:

In motion preferences, you can specify to select assigned parts whenever a motion file or path is selected in the **Paths** view.

Overview of the motion file status

The status of a motion file is indicated by its check box in the **Paths** view.

This check box state	Indicates this
	The motion file is open but not loaded. Select the check box to load the file into memory.
	The motion file or path is loaded but not visible in the Viewing window. Select the check box to display the motion data.
	The motion file or path is loaded and visible in the Viewing window. All occurrences are successfully resolved.
	The motion file is loaded and partially visible in the Viewing window. Some occurrences are not resolved.
	All of the part references in the motion file failed to resolve to the loaded product structure.

Update the product structure assignment

If a motion file fails to locate the expected root node of the assembly with which it is associated, you can attempt to manually establish a new assignment with appropriate product structure.

Tip:

You also can change motion assignments to new, but similar, product structure. This enables you to use motion files created for one model with different or revised models. The coordinate systems of the original and target parts must be aligned in order to produce the expected results.

1. In the **Paths** view, right-click a motion file or path and choose **Assignment**.
2. In the **Motion Assignments** dialog box, do any of the following:

To	Do this
Change the motion assignment	a. In the Viewing window, select an appropriate part or assembly. b. In the Assignments section, click Add Selected.

	c. In the Assignments section, select the product structure, and then click Set Selected .
Reconcile a failed assignment	a. In the Assignments section, select the failed product structure. b. In the Details section, click Re-target Root or Reconcile. c. (Optional) Create a report.

3. Click **Close**.

Overview of motion reconciliation

When a motion file cannot find expected product structure, the motion cannot play and you must create a new association with an appropriate part or assembly.

Motion reconciliation may be required in the following situations:

- If you want to replay the motion in a different context

If you want to replay motion data in a different context, the expected assembly node occurrence will not be found. Use the **Motion Assignments** dialog box to locate a suitable match.

- If no assembly matches are found

When the assembly node identifier in the motion file is not matched with any occurrences in the current assembly structure, you must manually select a new node.

- If some of the motion bodies fail to resolve

For cases where some, but not all, motion references resolve, you must manually resolve the failed occurrences. However, because some occurrences were resolved, most likely the expected root node was found, and using the motion reconcile tool to re-assign the failed references is unlikely to fix the problem. For cases like this, the general reconciliation tools are probably more suitable.

Use the Paths view

You can access much of the motion functionality from the **Paths** view.

Tip:

When you first load a motion file, some of the columns may not be visible in the **Paths** view. To turn on columns, in the area where column titles are displayed, right-click and choose the columns that you want to display. If columns are turned on but not visible, you need to **adjust the column width**.

In the **Paths** view, do any of the following:

To	Do this
Open a motion file	Right-click Motion Files and choose Open Motion File .
Create a new motion file	Right-click Motion Files and choose New Motion File .
Rename a motion file	Right-click a motion file and choose Rename .
Save a motion file	Right-click a motion file and choose Save or Save As .
Delete a motion file	Right-click a motion file and choose Delete . Note: You can use the Undo command to undo the last Delete action.
Add motion files to the play list	• (To add all open motion files) Right-click Motion Files and choose Add to Playlist. • (To add a single motion file) Right-click the motion file and choose Add to Playlist.
Access the motion preferences	Right-click Motion Files and choose Preferences .
Preview motion files	Right-click a motion file or path and choose Preview .
Select parts associated with motion files	Right-click a motion file or path and choose Select Parts . Tip: Specify in the motion preferences to select parts whenever a motion file or path is selected in the Paths view.
Create a swept volume	Right-click a motion file or path and choose Create a swept volume .
Change motion assignments	Right-click a motion file or path and choose Assignment .
Create a new path	Right-click a motion file and choose New Path .
Copy and paste a path	1. Right-click a path and choose Copy. 2. Right-click a path and choose Paste.
Edit a path	Right-click a path and choose Edit .
Add a new key frame to a path	Right-click a path and choose Insert Key Frame .

Optimize a path	Right-click a path and choose Optimize .
Delete a path	Right-click a path and choose Delete .
Edit a key frame	Right-click a key frame and choose Edit .
Pick a key frame	Right-click a key frame and choose Pick .
Delete a key frame	Right-click a key frame and choose Delete .
Reset a key frame	Right-click a key frame and choose Reset .
Lock a key frame	Right-click a key frame and choose Lock .

Optimize motion files

Reduce the number of path key frames before saving motion files.

1. On the **3D Motion** toolbar, click **Preferences** .

-or-

In the **Paths** view, right-click **Motion Files** and choose **Preferences**.

2. In the **Motion Preferences** dialog box, in the **Minimum percentage of key frames to reduce during path optimization** box, type a value between 0 and 100 for the lowest amount of key frames to remove.

Type a value of 0 for the application to determine the amount of key frames to reduce.

Type a value greater than 0 to eliminate additional key frames. This impacts the quality of the optimized path.

Tip:

Use **Motion Preferences** to specify to optimize motion files automatically when you load them.

3. Click **OK**.
4. In the **Paths** view, right-click a motion path and choose **Optimize**.

Save motion files

Save your motion information in the VFM or PLM XML format.

1. In the **Paths** view, select a motion file.

- On the **3D Motion** toolbar, click **Save Motion File** .

-or-

In the **Paths** view, right-click the motion file and choose **Save**.

Note:

Choose **Save As** if you want to save a motion file as a different type.

- In the **Select motion files** dialog box, browse to a location and type a name for the file, then choose one of the following file types:

Use	To do this
Motion Files (*.vfm)	Save motion data in the VFM format, which does not save key frame information.
PLM XML Motion Frame (*.plmxml)	Save motion data in the PLM XML Motion Frame format, which does not save key frame information.
PLM XML Motion Keyframe (*.plmxml)	Save motion data in the PLM XML Motion Keyframe format, which preserves key frame information.

- Click **Save**.

Note:

References to motion files are saved when you save your work as PLM XML (if you set the Save extended 3D content into PLMXML preference). If you have unsaved motion files, a dialog box appears asking if you want to save the changes to a file.

Converting .xmo and .res motion files

Overview of converting motion files

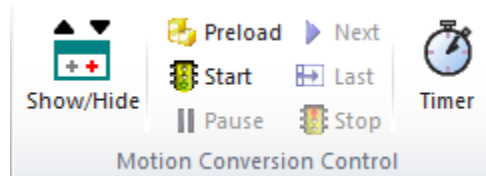
Convert .xmo motion files and Adams Results Files (.res) to the VFM motion format.

You can **automate the selection of the mapping file for conversion of .res files**.

Note:

- You must associate an .xmo or .res motion file with a part or assembly before you can save it as a VFM motion file.

- The data in an .xmo motion file must be properly structured.
- The ADAMS conversion feature, which converts RES files to the VFM motion file format, requires the Professional or Mockup product configuration, as well as an additional license.
- To activate the ADAMS conversion control features, you must set the environment variable **TCVIS_ADAMS_AUTOCONVERT** to **True**.



Tip:

You can automatically convert .xmo and .res motion files only to the VFM motion file format. However, once a file is converted to the VFM format, you can open it in the application and save it as a PLM XML motion file.

Convert .xmo files

1. Open the 3D model that is associated with the .xmo file that you want to convert.
2. Choose **Actions**→**Motion**→**Convert Motion Files**.
3. In the **File name** section of the **Select motion file to convert** dialog box, type the file name, or browse to the file.
4. Click **Open**.
5. In the Viewing window or in the **Assembly** view, select a part or assembly, and then, in the **Associated Part or Assembly** section of the **XMO File Conversion** dialog box, click **Capture Selected**.

Note:

You can associate an .xmo motion file only with a single part or assembly. You cannot associate an .xmo file with a collection of parts that do not form an assembly.

6. In the **Output VFM File** section, click **Browse**, navigate to the directory where you want to save the VFM file, type a name for the file, and then click **OK**.
7. (Optional) Select the **Open VFM file after conversion** check box to load the converted VFM file into the motion playback system automatically.

8. Click **OK**.

Convert .res files

1. Open the 3D model that is associated with the .res file that you want to convert.
2. Choose **Actions→Motion→Convert Motion Files**.
3. In the **File name** section of the **Select motion file to convert** dialog box, type the file name, or browse to the file.
4. Click **Open**.
5. In the **Associated Adams Model File** section of the **RES File Conversion** dialog box, click **Browse**, navigate to the Adams Model File (ADM) that you want to associate with the results data, and then click **OK**.

Note:

If you chose to **use a loaded mapping**, the file name and path already appear in this section.

6. In the **Output VFM File** section, click **Browse**, navigate to the directory where you want to save the VFM file, type a name for the file, and then click **OK**.
7. (Optional) Select the **Open VFM File after conversion** check box to load the converted VFM file into the motion playback system automatically.
8. Click **Next**.
9. In the **Adams Model Components** section, select an Adams Model Component from the list.
10. In the Viewing window or in the **Assembly** view, select a part or assembly, and then click **Capture Selected**.

Note:

You can associate each model component in an Adams Model File only with a single part or assembly. You cannot associate an Adams Model Component with a collection of parts that do not form an assembly.

11. (Optional) To disassociate a part from an Adams Model Component, select the name of a geometry association in the **Associated Part or Assembly** section, and then click **Remove Association**.
12. Click **Finish**.

13. If you chose the option to save mappings when you create them, the **Save As** dialog box appears. Save the mapping file with the desired name and location, and click **Save**.

Automate the conversion mapping file selection for .res files

A conversion mapping file correlates motion data between the Adams version and the VFM motion format of a model. A mapping file can be saved when you convert a .res file, and then you can reuse the mapping for subsequent conversions.

1. Create the mapping file to be used for converting .res files in the current session:
 - a. Choose **Actions→Motion→ADAMS Auto Convert**, and click **Save Mapping** so that a check mark appears next to it.

Now when you convert .res files during the current session, a dialog box appears that enables you to save the mapping file.
 - b. Convert a .res file, saving the mapping when the **Save As** dialog box appears.
2. Load the mapping file that you want to use for file conversions in the current session:
 - a. Choose **Actions→Motion→ADAMS Auto Convert→Open Mapping File**.
 - b. Browse to and select the Adams Motion Mapping file (.mmf) you want to use, and then click **Open**.
3. To use a loaded mapping file to convert .res files, choose **Actions→Motion→ADAMS Auto Convert**, and click **Use Loaded Mapping** so that a check mark appears next to it.

For each subsequent conversion you perform in the current session, the mapping file that is opened is used for the conversion. In the **Select motion file to convert** dialog box, the **Associated Adams Model File** option is automatically filled in.

To use a different mapping file, repeat step 2 above to load a different file.

.xmo motion file specifications

.xmo motion data is stored in a key frame structure. Each key frame provides data to determine the position of the associated part or assembly at that frame. The motion data stored in the file can be presented in two formats: absolute and relative.

If the file format is absolute, the key frame data represents the absolute position of the associated part or assembly. The position information in the data file is assumed to be the position of the part or assembly coordinate system as defined in the DirectModel (.jt) data file. The position of this coordinate system should be available from the authoring CAD package. If the position information is not the

position of the part or assembly coordinate system, the motion visualized will not be the expected results.

If the file format is relative, the key frame data represents the change in position between two absolute positions. In order to ensure that playback of .xmo files is deterministic, the change information will be assumed to indicate the motion of the associated part or assembly from the default part of the assembly position as specified in the DirectModel data file. At load time the default part or assembly position will be used to convert the relative motion data into a sequence of deterministic absolute positions.

Note:

- The .xmo file format does not specify the parts associated with the motion data. This allows you to import motion data without the need to understand the CADID part representation. Once an .xmo file is loaded, you must associate it with a part or assembly.
- The radix specifier for .xmo files is the decimal point (.).

An .xmo file is comprised of four elements: Motion, Frame, Quaternion, and Translation:

```
<Motion type="absolute"> <Frame> <Quaternion e1="1.0" e2="0.0" e3="0.0"
e4="0.0" /> <Translation x="0.0" y="0.0" z="0.0" /> </Frame> </Motion>
```

These elements are described in the following table:

Use this element	To do this
Motion type	This element is the root object of the document. It defines the outermost container of the motion data contained in the file, and can be Absolute or Relative. The Motion element can contain any number of Frame elements.
Frame	The Frame element defines the key component used in the execution of the motion file. A Frame contains one Quaternion element, and one Translation element.
Quaternion	This element defines the rotation information associated with a Frame element.
Translation	This element defines the translation information associated with a Frame element.

Specify motion preferences

1. On the **3D Motion** toolbar, click **Preferences** .

-or-

In the **Paths** view, right-click **Motion Files** and choose **Preferences**.

2. In the **Motion Preferences** dialog box, adjust any of the following settings:

Use this option	To do this
Add to the play list when opened	When a motion file is loaded, add it to the play list.
Optimize on load	When a motion file is loaded, reduce the number of key frames it contains.
Perform deep optimize	When a motion file is loaded, maximally reduce the number of key frames it contains.
Use current part position as path origin on load of VFM file	Start the part motion defined in a VFM file from the current part position. If this option is not selected, then before the start of motion, parts are restored to the position they were in when the VFM file was created.
Save VFM file transforms as	Specify how to save transformations in VFM files. Choose from • Global Delta: This stores transformations as global delta matrices with respect to the parent assembly. Use this setting if you work with inserted models and mixed measurement units. • Global Position: This stores transformations as absolute global positions. Use this setting if you are creating swept volumes. Tip: If you want to use an existing VFM file that contains transformations stored as global delta matrices to create a swept volume, load the VFM file and then save it again using the Global Position preference.
Use selected parts to define path origin and assignment	When creating a path, automatically assign the path to selected parts.
Default number of steps for key frames	Type the number of intermediate steps between key frames. Use a higher number for smoother motion.
Minimum percentage of key frames to reduce during path optimization	Type a value between 0 and 100 for the minimum amount of key frames to delete. Type a value of 0 for the application to determine the amount of key frames to reduce. Type a value greater than 0 to eliminate additional key frames. This impacts the quality of the optimized path.
Show step frames on paths	Display step frames in the Viewing window.

Show axis of rotation for selected key frame	Display the axis of rotation in the Viewing window.
Enable path manipulator	Enable the path manipulator.
Lock all key frames during edit	Prevent existing key frames from being edited.
Select assigned parts when Motion File or Path is selected	Select parts associated with a motion file or path in the 3D Viewing window.

Recording movies

Overview of movie capture

Capture video as you manipulate the contents of the 3D Viewing window. Save your movie as an .avi file playable with most standard software video players.

You can:

- Create a movie file.
- Create a movie file from snapshots.
- Set movie capture preferences.

Create a movie file

Record an .avi movie of the active contents of the Viewing window.

Tip:

If you enable anti-aliasing in the **3D viewing preferences**, your movie captures also will be anti-aliased. Enabling this option will provide finer image quality in your movies, but at the cost of longer rendering times.

1. On the **3D Movie Capture** toolbar, click **Record** .
2. In the 3D Viewing window, **move the camera** or otherwise manipulate your model.

As you manipulate the model in the Viewing window, the scene is captured as the movie file.

3. On the **3D Movie Capture** toolbar, click **Stop** .

4. In the **Save As** dialog box, browse to the directory where you want to save your file, name the file, and then click **Save**.

If the movie capture preferences are set to compress the file, the **Video Compression** dialog box appears.

5. Adjust the compression settings, if necessary, and then click **OK**.

The movie is saved.

Create a movie file from snapshots

1. **Capture snapshots**, and arrange them in the appropriate order.
2. On the **3D Movie Capture** toolbar, click **Preferences** .
3. In the **Movie Capture Preferences** dialog box, select the **Snapshot Interpolate Frames** check box.
4. In the **Snapshot Interpolate Frames** box, type the number of frames to interpolate between snapshots.
5. Click **OK**.

6. On the **3D Movie Capture** toolbar, click **Record** .
7. In the **Snapshots** view, right-click the first snapshot and choose **Cycle**.

The movie records as the model view cycles through the snapshots.

8. On the **3D Movie Capture** toolbar, click **Stop** .
9. In the **Save As** dialog box, browse to the directory where you want to save your file, name the file, and then click **Save**.

If the movie capture preferences are set to compress the file, the **Video Compression** dialog box appears.

10. Adjust the compression settings, if necessary, and then click **OK**.

The movie is saved.

Set movie capture preferences

You can set a variety of options for recording movie files.

Note:

By default, a frame is recorded every two seconds, the playback frame rate is set to five frames per second, and the movie is compressed.

1. On the **3D Movie Capture** toolbar, click **Preferences** .
2. In the **Movie Capture Preferences** dialog box, adjust any of the following settings:

Change this setting	To do this
Record a Frame Every	Specify how often frames are recorded. You can type a number between one and 60. Choose between Frames or Seconds. Example: If you type a value of two and select Seconds, the scene is captured every two seconds. If you select Frames, the scene is captured every second time the scene is redrawn in the Viewing window.
Playback Frame Rate	Specify the playback frames per second. You can type a number between one and 30.
Snapshot Interpolate Frames	Record the movie file from a series of snapshots. You can specify how many frames to use when transitioning between snapshots. A greater value results in a smoother transition, while a lesser value results in a more abrupt shift between snapshots.
Create Movie as Series of Bitmaps	Render the movie frames as a series of bitmaps (.jpg, .png, .tiff, or .bmp). The appropriate frame number is appended to the name of each generated file.
Use Compression	Compress the movie to a smaller file size using standard Video for Windows compression. By default, movies are compressed.
Use DirectShow	Compress the movie to a smaller file size using a DirectShow compliant codec present on your machine. Choose a codec in the DirectShow Compressor list. Note: Available compression codecs vary from machine to machine, depending upon whatever video-related software is installed. For information on a particular codec, contact the company that created it.

	Specify the movie dimensions. Choose from either NTSC or IMAX standard movie sizes. - From the Output Size list, select a movie size. (If you selected Custom) In the Width and Height boxes, type the dimensions you want for the movie. (Optional) From the Default Size list, select a resolution. Note: The choices available are dependent upon your Output Size selection. (Optional) To maintain the ratio of image height to width, click the Keep Aspect Ratio check box.
Output Size	
Temp Directory	Tip: To specify a different temp directory, in the Temp Directory box, type a file path, or click Browse, navigate to a directory, and then click OK.

- Click **OK**.

When you record your movie, the file is recorded and saved as you have specified.

12. Changing part appearance

Overview of part appearance

There are many ways to change the appearance of 3D geometry. You can:

- Render 3D geometry as **shaded, with tessellation lines, or with feature lines**.
- Show or hide 3D wireframe **construction geometry**.
- Render all solid geometry as transparent to **reveal wireframe geometry**.
- Use the **Quick Color Tool** to quickly change the color and transparency of parts and part faces.
- Apply a variety of **predefined visual effects** to your 3D models.
- Use the Appearance Editor to create **materials** that consist of color, shading, textures maps, bump maps, and environment maps.
- Add **light sources** to the Viewing window.
- Manage materials, images, texture coordinates, and lights with the **Appearance Palette**.
- Use the **assembly** to control appearance attributes.
- View **assembly appearance icons** in the assembly tree.
- Apply a **color range** to a model.
- Insert **2D images as flat pieces of 3D geometry** into the 3D scene.
- Specify Appearance Editor **preferences**.

Working with display modes

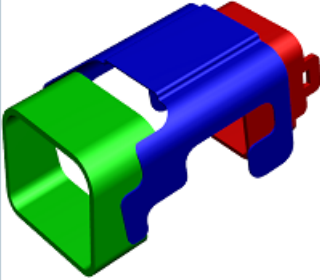
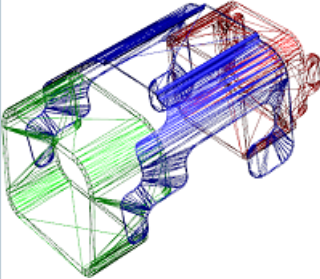
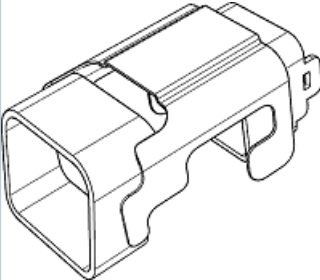
Display modes overview

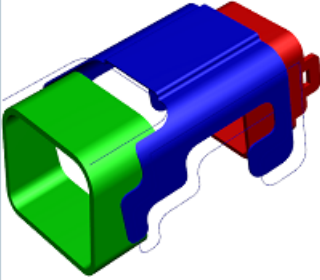
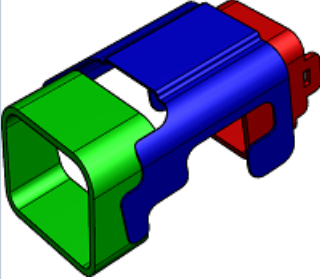
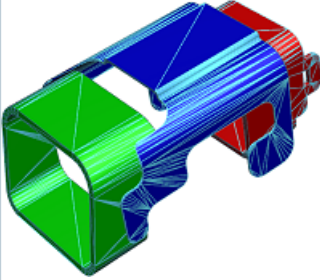
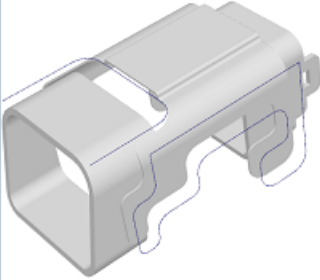
Change display modes to alter the appearance of geometry in the 3D Viewing window. Choose from the **Shaded**, **Tessellation Lines**, and **Feature Lines** rendering styles. Show or hide wireframe construction geometry. You can change the display mode of selected parts or the entire model. You can apply more than one mode. For example, display as shaded with tessellation lines. You can also render all solid geometry as transparent to reveal wireframe geometry.

Use the **3D Display Modes** toolbar to enable different display modes while working in the Viewing window. Use the **3D viewing preferences** to change the display mode of the active view and also to set the default display mode.

Note:

Part display mode states are saved in snapshots, session files, and .plmxml files with extended 3D content.

Display modes	Appearance in Viewing window
Shaded	
Construction Geometry	
Tessellation Lines Display the edges of polygons. Tessellation lines hidden by other geometry can be viewed as normal, semi-transparent, or invisible.	
Feature Lines Display edges as generated from feature angles or B-Rep data. Feature lines hidden by other geometry can be viewed as normal, semi-transparent, or invisible.	

Display modes	Appearance in Viewing window
Shaded and Construction Geometry	
Shaded and Feature Lines	
Shaded and Tessellation Lines	
See-Thru Solids Display all solid geometry as transparent to reveal wireframe geometry.	

Change the default display mode

The default display mode affects all parts except those with part-specific display modes.

You can change the default display mode from the **3D Display Modes** toolbar or the 3D Viewing preferences.

1. On the **3D Display Modes** toolbar, click one or more of the following display modes:

- Draw Shaded Model 

- Display Construction Geometry 

Tip:

When using **Display Construction Geometry**, use the **Construction Geometry On Top**  option to show internal construction geometry on top of the model.

- Draw Tessellation Lines 

- Draw Feature Lines 

Tip:

To pick through parts displayed in the tessellation or feature lines display modes, on the **3D Selection** toolbar, click **Pick Through Unshaded Parts** .

- If you choose the tessellation lines or feature lines display modes, you can display lines hidden by other geometry as visible , ghosted , or invisible .

Apply display modes to individual parts

- Select** the parts.
- On the **3D Display Modes** toolbar, click one or more of the following display modes:

- Draw Shaded Model 

- Display Construction Geometry 

- Draw Tessellation Lines 

- Draw Feature Lines 

Note:

All parts displayed in the tessellation lines and feature lines display modes share the same hidden lines setting (visible, ghosted, or invisible).

Reset display modes assigned to parts

On the **3D Display Modes** toolbar, click **Clear display modes** .

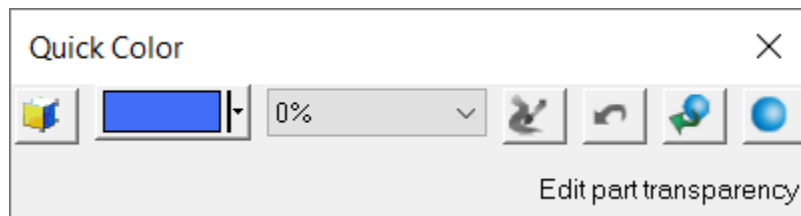
Note:

- Clearing the display mode is applicable for parts with unique display mode assignments. To change the default display mode, ensure that no parts are selected, and choose another display mode setting. Save your work session before resetting the display modes. You cannot undo a display mode reset.
- Clearing the display mode is also applicable for the model View Style set by PMI.

Using Quick Color Tool to change color and transparency

Overview of the Quick Color Tool

Use the Quick Color Tool to quickly change the color and transparency of parts and **sub-part geometry** (shapes and faces). You can also copy color and transparency settings from a part, shape, or face and then paint them on to different parts, shapes, or faces.



Note:

- You must have V9.1 or later JT data to assign materials to faces. With pre-V9.1 JT files, picking a face picks the parent shape.
- If you use Quick Color to change the color of a part, you cannot then change the color of individual faces on the part. You can restore the original color of the part and then change selected faces.
- The JT file must have B-Rep data for you to apply color to faces.

Tip:

After using the Quick Color Tool to modify the appearance of your model, you can use a session file or PLM XML to save your changes.

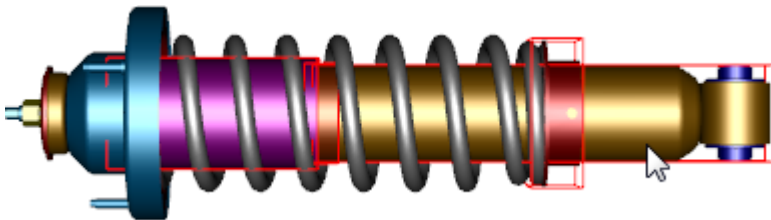
Use Quick Color to change color and transparency of parts

Use the **Quick Color Tool** to quickly change the color and transparency of parts.

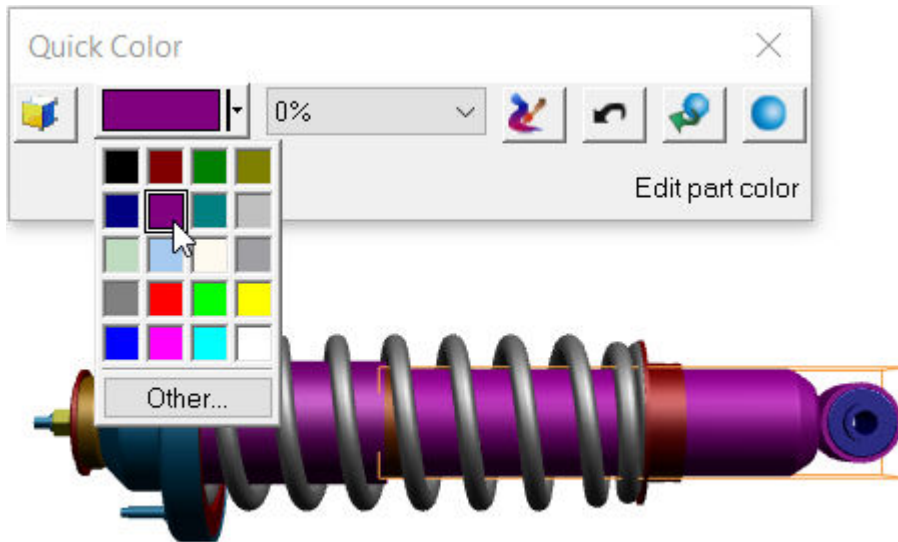
The following apply:

- If you use Quick Color to change the color of a part and the part's assembly has an overriding material, the part assignment will be made but the part will not change color due to the overriding material on the assembly.
- When the **Material** column is turned on, additional information about overriding materials is gathered. In this situation, if you use Quick Color to change the color of a part and the part's assembly has an overriding material, a **Quick Color Warning** dialog displays indicating that the override material assignments must be removed before editing the parts.
- If you use Quick Color to change the color of a part, you cannot then change the color of individual faces on the part. You can restore the original color of the part and then change selected faces.

1. On the **3D Appearance** toolbar, click **Quick Color Tool** .
2. In the **Quick Color** dialog box, deselect the **Edit Faces**  button if it is active.
3. In the Viewing window or the assembly tree, select the part to change (or press the Ctrl key and select the multiple parts).

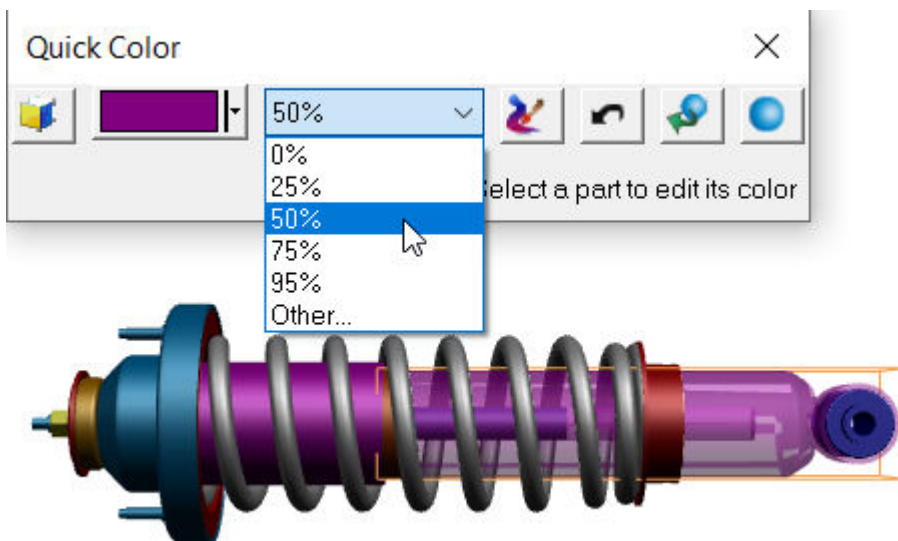


4. In the **Quick Color** dialog box, choose a color from the color well.



The color of the selected parts changes.

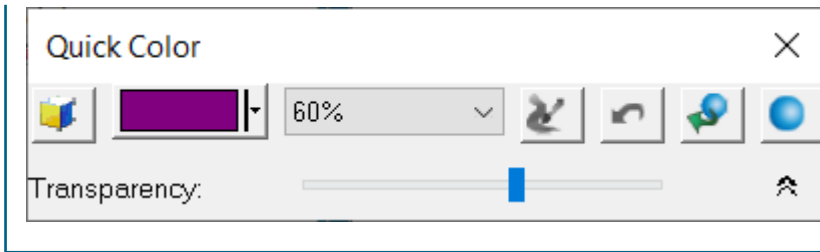
5. From the transparency list, select a level of transparency.



The level of transparency of the selected parts changes.

Tip:

Select **Other** from the transparency dropdown list to adjust the level of transparency with a slider. Drag the slider left and right to make your parts more or less transparent. The transparency percentage that you specify with the slider is added to the list of transparency values in case you want to use the same setting with other parts.



- Click the  in the corner of the **Quick Color** dialog box to close it.

Use Quick Color to change color and transparency of faces or shapes

Note:

You must have V9.1 or later JT data to assign materials to faces. With pre-V9.1 JT files, picking a face picks the parent shape.

The JT file must also have B-Rep data for you to apply color to faces.

- On the **3D Appearance** toolbar, click **Quick Color Tool** .
- In the **Quick Color** dialog box, click the **Edit Faces** button  so that it is activated.
- Do one of the following:

To	Do this
Change faces	In the Viewing window, select the face to change (or press the Ctrl key and select the multiple faces).
Change shapes	Do one of the following: - Hold Shift and click a shape that you want to include in the material assignment (or press Ctrl+Shift and select multiple shapes). - Double-click a shape.

- In the **Quick Color** dialog box, choose a color from the color well.

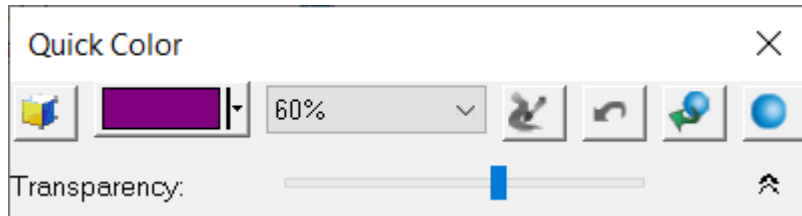
The color of the selected faces or shapes changes.

- From the transparency list, select a level of transparency.

The level of transparency of the selected faces or shapes changes.

Tip:

Select **Other** to adjust the level of transparency with a slider. Drag the slider left and right to make your faces more or less transparent. The transparency percentage that you specify with the slider is added to the list of transparency values in case you want to use the same setting with other parts or faces.

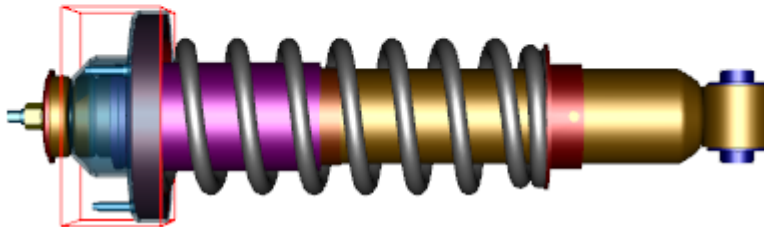
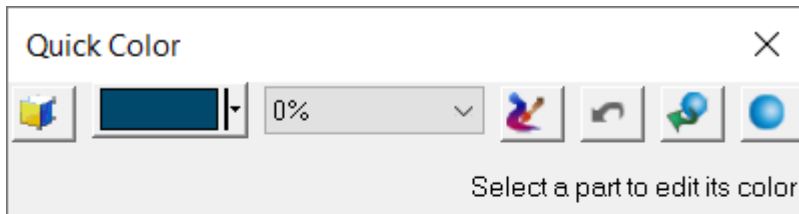


- Click the  in the corner of the **Quick Color** dialog box to close it.

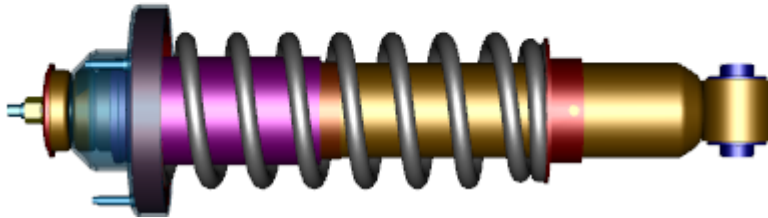
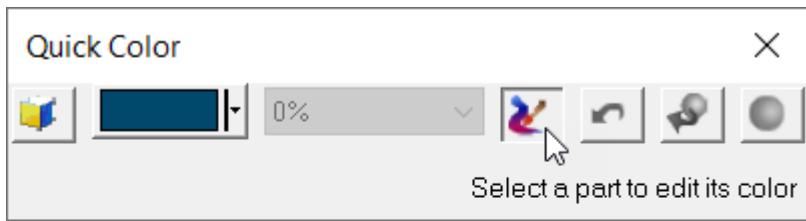
Copy color settings to other parts, faces, or shapes

- On the **3D Appearance** toolbar, click **Quick Color Tool** .
- In the Viewing window or the assembly tree, select a source part, face, or shape with appearance settings that you want to copy.

The color well and transparency list display the settings from the selected part.



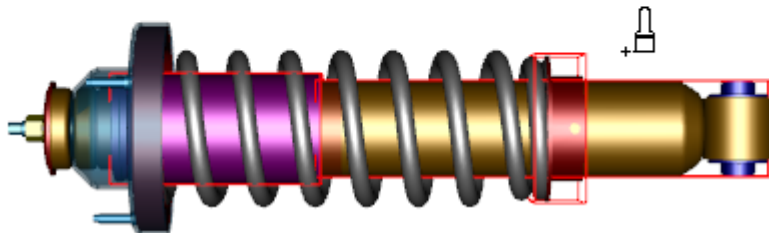
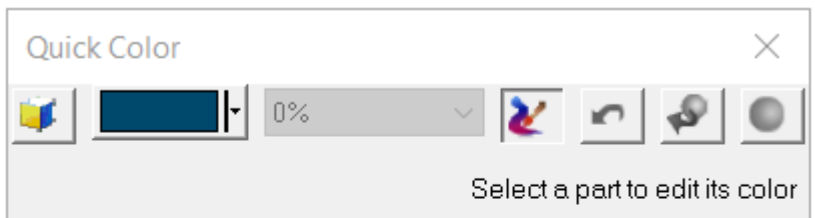
- Click the **Painter** button .



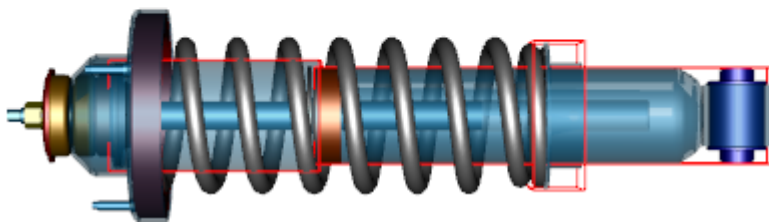
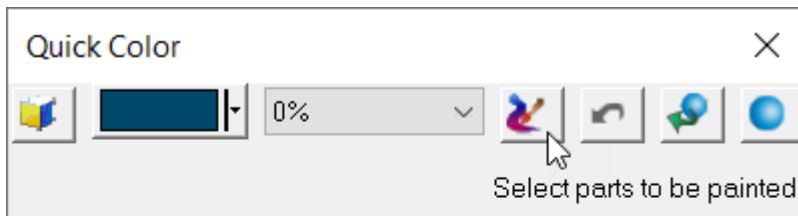
The mouse pointer turns into a paint brush  and the source part is deselected.

4. Do one of the following to select the parts, faces, or shapes to paint:

To	Do this
Paint parts	- In the Quick Color dialog box, make sure Edit Faces is not active - In the Viewing window, select the part to paint (or press the Ctrl key and select the multiple parts).
Paint faces	- In the Quick Color dialog box, click the Edit Faces button  if it is not already active. - In the Viewing window, select the face to paint (or press the Ctrl key and select the multiple faces). Note: You must have V9.1 or later JT data to assign materials to faces. With pre-V9.1 JT files, picking a face picks the parent shape.
Paint shapes	- In the Quick Color dialog box, click the Edit Faces button  if it is not already active. - Do one of the following: - Hold Shift and click a shape to paint (or hold Ctrl+Shift and select multiple shapes). - Double-click the shape to paint.



5. Click the **Painter** button again .



The selected parts, faces, or shapes change to match the source part.

Tip:

Click the **Undo** button  in the **Quick Color** dialog box to disregard your last action. The list of Undo actions is reset if the **Quick Color** dialog box is closed.

6. Click the  in the corner of the **Quick Color** dialog box to close it.

Note:

You cannot copy a color to parts, faces, and shapes in one operation.

Use Quick Color to restore original color and transparency

- On the **3D Appearance** toolbar, click **Quick Color Tool** .
- Do one of the following:

To restore original color and transparency of	Do this
Selected <i>parts</i>	- In the Quick Color dialog box, deselect the Edit Faces  button if it is active. - Select parts to restore. - Click the Restore button .
Selected <i>faces</i>	- In the Quick Color dialog box, click the Edit Faces button  if it is not already active. - Select faces to restore. - Click the Restore button .
Selected <i>shapes</i>	- In the Quick Color dialog box, click the Edit Faces button  if it is not already active. - Do one of the following to select shapes to restore. - Hold Shift and click the shape (or hold Ctrl+Shift and select multiple shapes). - Double-click the shape. - Click the Restore button .
All <i>visible</i> parts, faces, and shapes	- Make sure no parts are selected. - Click the Restore button .

- Click the  in the corner of the **Quick Color** dialog box to close it.

Working with True Shading

True Shading overview

True Shading is a collection of predefined visual effects that you can use with your models to create compelling 3D scenes for product demonstrations or high quality illustrations. You can apply various surface finishes, such as metallic or glossy, to your entire model or individually to single parts and subassemblies. Additional effects include global reflections, background images, and floor effects, such as a grid, a drop-shadow, and a model reflection.



Understanding True Shading

True Shading is a visual display mode which you can use to quickly change the appearance of 3D models. When you turn on True Shading, the default settings for the various effects are applied automatically to your model in the Viewing window.

True Shading graphics hardware requirements

Only certified graphics cards that fully support OpenGL 3.3 or greater are capable of utilizing all of the advanced features of Teamcenter lifecycle visualization, including True Shading. When a graphics card is not capable of rendering an advanced visualization feature, the visual effect is silently omitted with no ill effects.

How True Shading affects other viewer settings

When you turn on True Shading, the viewer settings listed below are automatically adjusted to enhance the appearance of your geometry. When you turn off True Shading, the previous settings are restored.

- If the viewing mode is set to **Orthographic**, it is changed to **Perspective**.

- The ambient light factor is set to **0.21**.
- The **Cast shadows defined by light sources** performance setting is enabled.
- If the selection mode is set to **Material Color**, it is changed to **Bounding Box**.

Saving the True Shading state

You can use session files and PLM XML to save and restore all of your True Shading settings, including occurrence specific material assignments. If the True Shading display mode is enabled when you save a session file or PLM XML file, it is also enabled when you open either file type to restore the session.

True Shading material assignments are also preserved in exported JT models.

Capturing the True Shading state in 3D snapshots

You can use 3D snapshots to capture and restore your True Shading settings, provided the **True Shading snapshot preference** is turned on.

If you activate a snapshot created with the **True Shading** snapshot preference turned off, there is no impact on the current True Shading state in the Viewing window. If True Shading is off, it remains off; if it is on, it remains on. The True Shading asset is not present in the snapshot, so it has no effect on the True Shading state of the session.

If you activate a snapshot created with the **True Shading** snapshot preference turned on, the behavior can vary in the following ways:

- If the True Shading display mode is off when you activate the snapshot, but on when the snapshot was created, it is turned on; if you subsequently activate a snapshot captured when the True Shading state was disabled, the display mode is turned off.
- If the display mode is on when you activate the snapshot, it will stay on, even if the True Shading state was disabled when the snapshot was created.

True Shading limitations

The following limitations apply when working with True Shading:

- You cannot use the True Shading display mode at the same time as the Concept appearance system or the Quick Color Tool. In addition, True Shading materials supersede any existing materials that are associated with 3D models, including materials created with the Concept appearance system. Global wash materials are an exception to this rule because they blend with the color and transparency settings of the model's underlying materials.

- For any particular 3D document, you cannot work with True Shading in multiple viewing windows. You can open multiple instances of a single 3D model, and work with different True Shading effects in each document, but if you create or clone a window, True Shading is disabled.
- True Shading effects may not display as expected with **JtSimplification data**.
- Surface analysis effects are disabled.
- The Part Edit functionality is disabled.
- True Shading effects are applied to full parts only. Occurrence specific materials cannot be applied to individual **shapes** or surfaces.
- Annotations, such as measurements or markups, are not included in the floor reflection or shadow.

True Shading materials

The True Shading display mode offers global and occurrence specific materials. Global materials change the appearance of the entire 3D model; occurrence specific materials affect individual parts and subassemblies.

A global material is always active in the 3D scene. When an occurrence specific material is applied to a part or subassembly, it overrides the global material, but if the occurrence specific material is removed, the part or subassembly returns to the style of the active global material.

Most True Shading materials are available for use on a global and occurrence specific basis, with these exceptions:

- Washes, which are available only as global materials
- Texture finishes, which are available only as occurrence specific materials

Note:

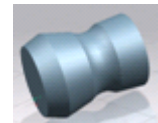
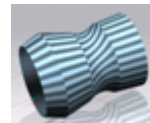
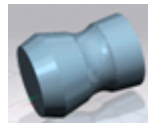
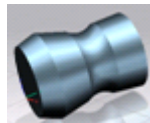
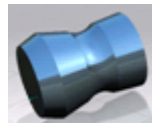
Some materials, such as **Brushed Chrome**, do not display global reflections. These materials are marked below with the following symbol: .

Available True Shading materials are shown below.





1 Washes
(global materials only)



Shiny Metal Wash



Brushed Metal Wash



Glossy Plastic Wash



Surface Analysis Wash



Low Sheen Plastic Wash

2 Glossy plastic finishes



Red Glossy Plastic



Blue Glossy Plastic



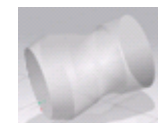
Green Glossy Plastic



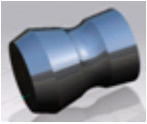
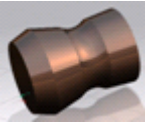
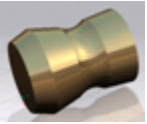
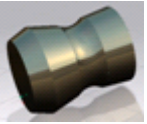
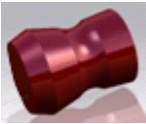
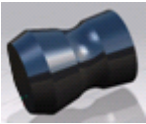
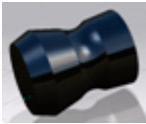
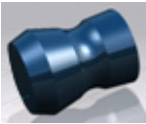
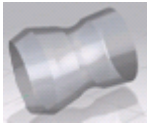
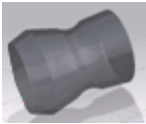
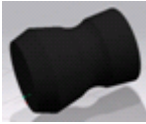
Gray Glossy Plastic



Black Glossy Plastic

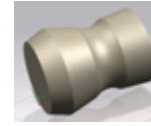
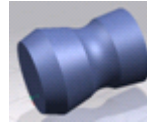
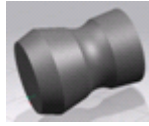


Clear Plastic

		Brown Glossy Plastic	Yellow Glossy Plastic	Teal Glossy Plastic	White Glossy Plastic
3	Metal finishes				
					
		Chrome	Copper	Gold	Brass
4	Brushed metal finishes				
					
		Brushed Chrome	Brushed Aluminum	Brushed Titanium	
5	Metallic paint finishes				
					
		Red Metallic Paint	Gun Metal Gray Paint	Black Metallic Paint	Blue Metallic Paint
6	Glass finishes				
					
		Clear Glass	Dark Glass		
7	Rubber finish				
					

Black Rubber

- 8 Texture finishes (occurrence specific materials only)



Medium Gray Texture



Dark Gray Texture

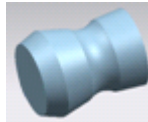


Blue Gray Texture



Tan Texture

- 9 Remove occurrence specific material (occurrence specific materials only)



Remove Material Assignment

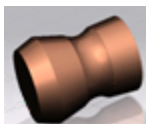
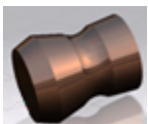
True Shading global reflections

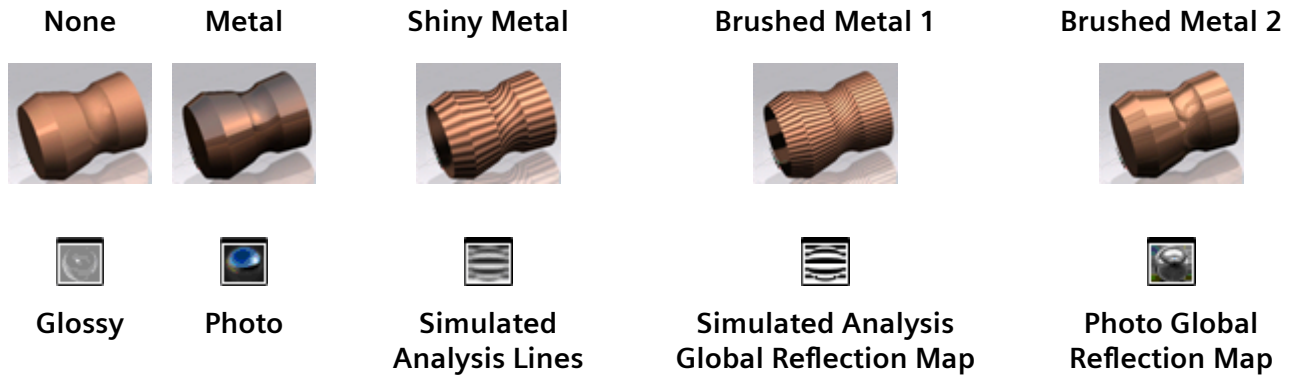
The True Shading display mode includes a variety of preset global reflections that you can apply to your entire 3D model. A global reflection creates the impression of a surrounding physical environment reflected upon the surfaces of the 3D model.

Note:

Some materials do not display global reflections.

Available global reflections are shown below.





Apply True Shading global materials

Use True Shading global materials to change the appearance of entire 3D models.



Tip:

Use True Shading occurrence specific materials to change the appearance of individual parts and subassemblies.

1. On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

2. Click **Global Materials** .

Tip:

If the **Global Materials** option is grayed out, True Shading materials is turned off. To turn on True Shading materials, click **Apply True Shading Materials** .

3. On the **Global Materials** tab of the **True Shading Editor**, select a material.

Apply True Shading occurrence specific materials

You can use True Shading occurrence specific materials to change the appearance of individual parts or subassemblies.



1. On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

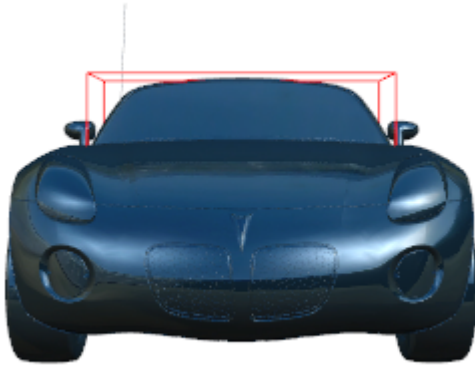
2. Click **Occurrence Specific Materials** .

Tip:

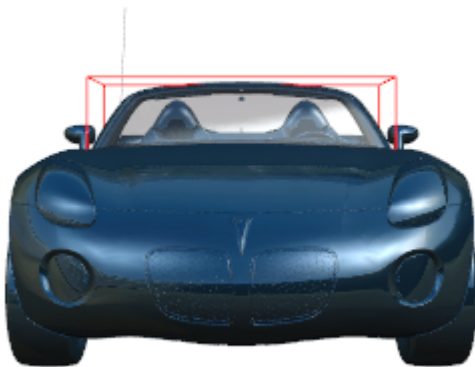
If the **Occurrence Specific Materials** option is grayed out, True Shading materials is turned off. To turn on True Shading materials, click **Apply True Shading Materials** .

3. Select a part or subassembly.

The geometry is highlighted.



4. On the **Occurrence Materials** tab of the **True Shading Editor**, select a material.



Remove True Shading occurrence specific materials

You can remove occurrence specific material assignments from individual parts or subassemblies. You can also remove all occurrence specific material assignments at once from your 3D model.

1. On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

2. Click **Occurrence Specific Materials** .

Tip:

If the **Occurrence Specific Materials** option is grayed out, True Shading materials is turned off. To turn on True Shading materials, click **Apply True Shading Materials** .

3. Do one of the following:

To	Do this
Remove an occurrence specific material assignment from a part or subassembly	- Select the part or subassembly. - On the Occurrence Materials tab, click Remove Material Assignment .
Remove all occurrence specific material assignments from a model	- Ensure that no parts or subassemblies are selected. - On the Occurrence Materials tab, click Remove Material Assignment .

The part, subassembly, or model is displayed using the currently active global material.

Apply True Shading global reflections

You can use a global reflection to create the effect of a mirrored surface on your 3D model.



- On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

- Click **Global Reflections** .

Tip:

If the **Global Reflections** option is greyed out, True Shading materials is turned off. To turn on True Shading materials, click **Apply True Shading Materials** .

- On the **Global Reflections** tab of the **True Shading Editor**, select a reflection.

Note:

Some materials do not display global reflections.

Turn True Shading materials on or off

You can control the display of True Shading materials and global material reflections independently of the other effects. When you deactivate **Apply True Shading Materials**, global materials, occurrence-specific materials, and global material reflections are no longer displayed. This enables you to retain the original appearance of your model while displaying other True Shading effects, such as a floor shadow.

On the **3D True Shading** toolbar, click **Apply True Shading Materials** .

Display True Shading background images

Enhance the appearance of your 3D scene with background colors, gradients, or preset images.



- On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

- Click **Background** .

3. On the **Background** tab of the **True Shading Editor**, do any of the following:

To	Do this
Use a preset color or gradient for the background	From the Background Type list, select one of the following:  Graduated Dark  Graduated Light  Dark  Light  Inherit Tip: Inherit uses the background color settings from the 3D viewing preferences, which enable you to create a gradient that can consist of up to four colors, one for each corner of the viewing window.
Use a preset image for the background	- From the Background Type list, select Image Background. - From the Background Image list, select one of the following:  Image 1  Image 2  Image 3  Image 4  Image 5

To	Do this
	•  Image 6
Choose a solid color for the background	- From the Background Type list, select Custom Solid. - For Top Color, click the color well and then select a color.
Create a gradient for use as the background	- From the Background Type list, select Custom Graduated. - For Top Color, click the color well and then select a color. - For Bottom Color, click the color well and then select a color.

Note:

True Shading background color settings are preserved in session files and are automatically restored when you reopen the session. If you do not want to overwrite your existing background color settings in the 3D viewing preferences, turn off True Shading before you save the session file.

Display True Shading floor effects

The True Shading floor is an invisible plane upon which you can display a grid, the model reflection, and the model shadow. Turn on any of these effects to make the floor visible in the Viewing window.



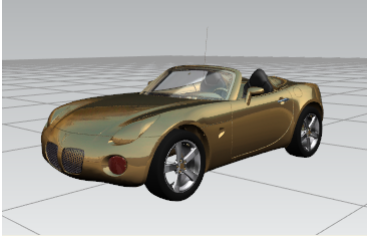
1. On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

2. Click **Show Floor** .

The floor is added to the 3D scene. It is visible only when at least one of the effects described below is active.

3. Turn on any of the following effects:

To	Do this
Display the True Shading floor grid	Click Show Grid  . 
Display the True Shading floor reflection	Click Show Floor Reflection  .  Note: The floor reflection is not visible when the camera is not on the same side of the floor as the geometry.
Display the True Shading floor shadow	Click Show Shadows  .

To	Do this
	 Note: For True Shading shadows to display, Cast shadows defined by light sources must be turned on in the 3D performance preferences. When you enter the True Shading display mode, this option is turned on automatically. However, when you change to a saved performance setting where this option is turned off, such as Performance Setting 3 (Model Exploration) on the 3D Performance toolbar, True Shading shadows become disabled. To display shadows again, select the Cast shadows defined by light sources check box in the 3D performance preferences. Tip: You can adjust the visual quality of the True Shading floor shadow on the Global page of the 3D performance preferences.

Change the True Shading floor position

Change the position of the True Shading floor in relation to your geometry in the Viewing window.

- 1. On the **3D True Shading** toolbar, click **True Shading** .

In the Viewing window, the appearance of the 3D model updates according to the default True Shading settings.

- 2. Click **True Shading Editor** .
- 3. On the **Floor** tab of the **True Shading Editor**, do any of the following:

To	Do this
Change the floor direction	From the Plane box, select one of the following: • XY • XZ • YZ • -XY • -XZ • -YZ • Custom Note: If you select Custom, click Pick, and then pick a part feature with which to align the floor.
Move the floor closer to or further from the model	Drag the Offset slider to the left or right. Note: When the floor is first turned on, its position is based upon the bounding box of the visible model. If you turn off the display of a part, the dimensions of the bounding box may change, which in turn may lead to a change in the floor position. To prevent the position of the floor from changing, select the Fix offset check box. If you turn off a part and want to update the floor position manually, click Update.

Working with materials

Materials overview

Materials are collections of visual properties that you apply to surface geometry to alter the appearance of your models. They can consist of color, shading, texture maps, bump maps, and environment maps.

There are three material types, basic, advanced, and SVM (Siemens Visual Material). Use basic materials when you want to change part color or shading only; use advanced and SVM materials when you want to use texture mapping, bump mapping, and environment mapping. SVM materials use textures that represent a real-life material, such as carpeting. Teamcenter lifecycle visualization automatically scales the SVM material so it has the correct look, independent of the model unit size.

Note:

Users who do not have SVM materials installed will receive an error message when opening a JT file that contains SVM materials, and the associated SVM materials may not render properly.

Create new materials

Use materials to change the color of geometry.

Tip:

You can also create new materials quickly by copying settings from existing materials.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
 2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
 3. In the **New Material** dialog box, choose a material type, and click **OK**.
- A thumbnail image of the material appears on the Appearance Palette.
4. In the **Colors** section, click the color wells to adjust the settings for each color type.
 5. (Optional) In the **Shading** section, adjust the Transparency, Shininess, and Reflectivity settings.

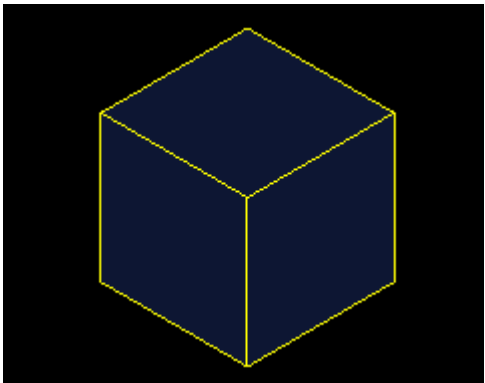
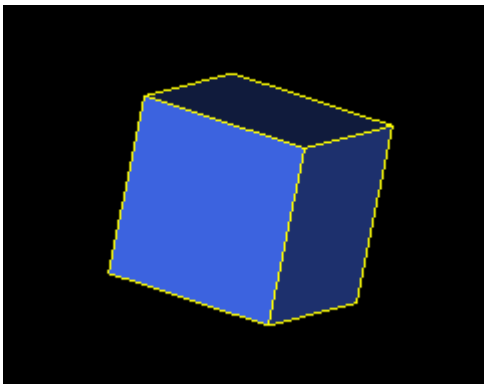
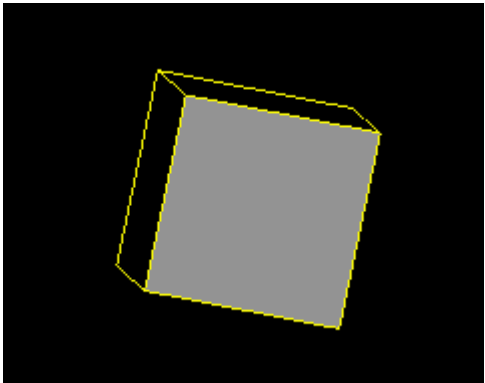
Reflectivity only affects how Environment Maps are rendered.

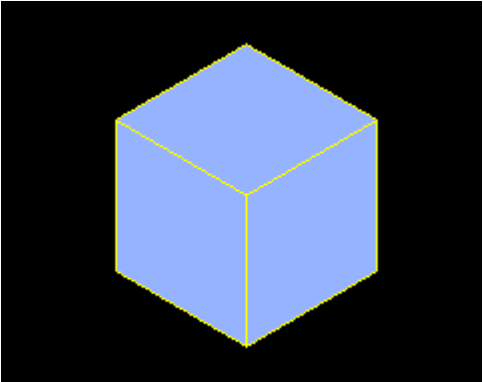
Changing the color and shading of geometry

Overview of using color

There are four related color types which collectively affect overall part color.

This color type	Produces this effect
Ambient	Even, unvarying color, which when isolated from the other color types is shown as unaffected by light sources.

This color type	Produces this effect
	Example: When viewing a material with an ambient color value of blue, and with the other color values set to "off" or black, the shade of blue on the surface of the geometry does not change, regardless of the position of the geometry in relation to the light source.
Diffuse 	Even color under direct light, tapering down to dimmer shades as the geometry changes in relation to the light source. Example: When viewing a material with a diffuse color value of blue, and with the other color values set to "off" or black, the shade of blue on the surface of the geometry changes, depending upon the position of the geometry in relation to the light source.
Specular 	The color of surface highlights on a part. Specular highlights resemble glare, or the most intense area of light cast upon an object and viewed from a particular angle. The effect of specular lighting changes based upon perspective and the position of the light source. Example: When viewing a material with a specular color value of white, and with the other color values set to "off" or black, the shade of white on the surface of the geometry changes, depending upon the position of the geometry in relation to the light source. If directly aligned "beneath" the light source, the shade appears white; if the position of the geometry or light source changes, the shade also changes, blending with the other color values as well as the color of the light source. Surfaces outside the scope of the light source exhibit no specular color.

This color type	Produces this effect
	Tip: For a realistic effect, specular color should approximate the color of the light source.
Emissive 	An even appearance of color, which when blended with ambient and diffuse color values resembles glow. This effect is limited to geometry with the material, and does not affect other parts in the Viewing window. Example: When viewing a material with an emissive color value of light gray, and with ambient and diffuse color values set to shades of blue, the geometry appears to produce light.

Use materials to change part color

Use new basic materials to change the color of geometry without modifying existing materials.

Tip:

In addition to creating and assigning new materials to your geometry, there are several additional methods of modifying the color of parts. If a material is already assigned to the part, and you do not need to preserve it, you can edit the existing material. This method is particularly useful because you can view the effects of your changes on your geometry in the Viewing window while you make them. However, if a material is assigned to other parts or you want to keep it for later use, you should clone the material or create a new one. To avoid modifying a material you want to keep, you can clone it or assign a different, existing material.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Material** dialog box, choose the **Basic** material type, and click **OK**.
A thumbnail image of the material appears on the Appearance Palette.
4. In the **Colors** section, click the **color wells** to adjust the settings for each **color type**.

- In the **Assignment** section, choose **Visible** parts, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select the parts or assemblies that you want to include in the material assignment.

- Click **Set**.

The material is assigned to the geometry in the Viewing window.

Change part shading

Use materials to change the shading of geometry. Shading options include transparency, shininess, and reflectivity.

Tip:

Create new materials quickly by **copying settings from existing materials**.

- On the **3D Appearance** toolbar, click **Edit Materials** .
- On the Material page of the Appearance Editor, in the **Management** section, click **New**.
- In the **New Material** dialog box, choose the **Basic** material type, and click **OK**.

A thumbnail image of the material appears on the Appearance Palette.

- In the **Shading** section, adjust any of these settings:

To	Do this
Make parts transparent	In the Transparency box, type a value between 0 and 1. -or- Drag the slider to the left or right. Note: A part with a transparency of 1 is invisible; a part with a transparency of 0 is opaque.

To	Do this
	In the Performance dialog box, select the Advanced Transparency check box to improve the quality of transparency in the Viewing window.
Make parts shiny	In the Shininess box, type a value between 0 and 1. -or- Drag the slider to the left or right. Note: A part with a shininess of 0 displays a large, washed-out highlight; a part with a shininess of 1 displays a small, well-defined highlight.
Reflectivity	Use the Reflectivity slider to determine the amount of reflection used when rendering an environment map.

- In the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the material assignment.

- Click **Set**.

The material is assigned to the geometry in the Viewing window.

Use the Color Editor

Use the Color Editor to choose colors from the HSV color palettes.

- Type values in the **HSV** boxes.

-or-

Click the color bars and drag the sliders.

-or-

Click the triangle area at the top of the Editor, and drag the pointer.

Note:

In the ring surrounding the triangle, adjust the position of the two sliders to change the color range displayed in the triangle. Use the cursor in the triangle to choose a color; moving the cursor horizontally changes the hue, and moving the cursor vertically changes the saturation. Use the color bar to the right of the triangle to adjust the luminosity, or relative degree of lightness or darkness, of the color range displayed in the triangle.

-or-

Click and hold the Eyedropper, move the mouse to a color anywhere on your screen, and release.

The right color box in the middle of the Editor updates to reflect the selected color.

2. Click **OK**.

Tip:

The two color boxes on the Color Editor allow you to preview two colors before making your final color selection. The left box contains the saved color, and the right box contains the active color.

When editing color values of a material that you have applied to a part, your color selections in the Color Editor are automatically previewed in the Viewing window. As you experiment with colors, use the buttons marked with the left and right arrows to switch between the saved color and the active color.

Using textures

Overview of texture layers

Texture maps are 2D images applied to the surfaces of 3D models. Common uses of texture maps include tiling a pattern across geometry to simulate surfaces such as brick or cloth, or applying a single instance of an image as a decal.

For texture mapping to have a visual effect on geometry, the geometry also needs texture coordinates. Each texture map layer has individual texture coordinates. When you create a texture map and assign a 2D image to a texture layer, the corresponding texture coordinates are automatically associated with the texture map. You can also manually create texture coordinates and associate them with any of the available texture layers.

A single part or assembly can have up to four layers of textures, which are applied one over the other, as well as four sets of texture coordinates, exactly one for each texture layer.

Note:

- Only texture layer 1 allows the full choice of texture coordinate types (planar, cubic, cylindrical, spherical). Texture layers 2 – 4 support only planar texture coordinates.
- Bump maps occupy their own unique layer, in addition to the four layers available for texture maps. However, bump maps always use the texture coordinates associated with texture layer 1.
- Environment maps use one of the four texture map layers. If your material includes an environment map, only three layers will be available for additional textures.

Load texture images

Load 2D raster images for use with textures.

Do one of the following:

To load an image	Do this
From the Appearance Editor	1. On the 3D Appearance toolbar, click Load Images . 2. On the Image page of the Appearance Editor, in the Management section, click New. 3. In the Choose image location dialog box, open your 2D image. A thumbnail image of the file is displayed on the Appearance Palette. The image also is displayed in the Preview section in the Appearance Editor. In the Information section, the following information is listed: • Type — The image format. Supported 2D raster formats are TIFF, JPEG, BMP, GIF, and PNG. • Format — The color channels of the image, in the order in which they are stored in the file. For example, "bgr" stands for blue-green-red. If an Alpha channel is present, it is displayed as an "a". For example, a blue-green-red file with an Alpha channel probably would be displayed as "bgra". • Dimensions — The image dimensions. Possible values are "1" and "2". Typically used for legends, one-dimensional images consist of a single line of horizontal pixels. They are repeated vertically in the image preview to better show the image colors.

To load an image	Do this
	• Size (pixels) — The number of horizontal and vertical pixels in the image.
From the Appearance Palette	1. On the 3D Appearance toolbar, click Image Palette . 2. On the Images page of the Appearance Palette, right-click an empty area, and select New. 3. In the Choose image location dialog box, open your 2D image. A thumbnail image of the file is displayed on the Appearance Palette. Tip: When you load an image onto the Appearance Palette, it is automatically selected.

Create texture maps

Use texture maps to apply 2D images to the surfaces of 3D models.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
 2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
 3. In the **New Material** dialog box, choose the **Advanced** material type, and click **OK**.
- A thumbnail image of the material appears on the Appearance Palette.
4. In the **Material** section, select the **Texture Map** check box.

The **Texture Map Layer 1** and **Texture Map Layer 2 to 4** sections become active.

Tip:

You can create up to four layers of textures. After creating the first texture layer in the **Texture Map Layer 1** section, create additional layers in the **Texture Map Layer 2 to 4** section.

5. In the **Texture Map Layer 1** section, select the **layer 1** check box.
6. Click the empty image box to bring up the Images page of the Appearance Editor, and then select an image.

-or-

Load a new 2D image, and then click the image box.

In the **Texture Map Layer 1** section of the Material page of the Appearance Editor, the 2D image appears in the image box.

7. (Optional) From the list to the right of the image box, change the **blending method**.
8. (Optional) Select the **Specular highlights on top** check box.
9. In the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the material assignment.

10. Click **Set**.

In the Viewing window, the texture map is assigned to the geometry.

Create bump maps

Use bump maps to make the surfaces of parts look as if they are bumpy or textured.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Material** dialog box, choose the **Advanced** material type, and click **OK**.

A thumbnail image of the material appears on the Appearance Palette.

4. In the **Material** section, select the **Bump Map** check box.

The **Bump Map** section becomes active.

Note:

Bump map textures always appear on texture layer 1.

5. Click the empty image box to bring up the Images page of the Appearance Editor, and then select an image.

-or-

Load a new 2D image, and then click the image box.

In the **Bump Map** section of the Material page of the Appearance Editor, the 2D image appears in the image box.

6. Choose one of the following bump map types:

Choose	If you want to do this
Absolute	Accept the default level of bumpiness.
Relative to normal vector	Adjust the degree of bumpiness. In the Bumpiness box, type a value. The higher the value, the bumpier the texture will appear. You can adjust the input range in the boxes below the slider. -or- Drag the slider to the left or right.

7. In the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the material assignment.

8. Click **Set**.

The material is assigned to the geometry in the Viewing window.

Note:

- A bump map texture reflects light differently from different angles. You may need to adjust your lighting to view bump maps correctly.
- When using a bump map with an environment map, select the **Use Clearcoat** option, located under the **Environment Map** section, to get the desired shiny effect.
- Not all graphics cards support bump maps.

Creating environment maps

Overview of environment maps

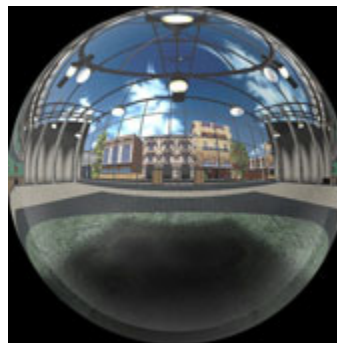
Environment maps, also known as reflection maps or reflectance maps, are textures that make 3D geometry look as if it reflects a surrounding environment. They work especially well when emulating the reflective properties of shiny surfaces such as chrome.

You can use two types of reflectance maps to enhance the appearance of your 3D models: static and dynamic environment maps.

Static environment mapping

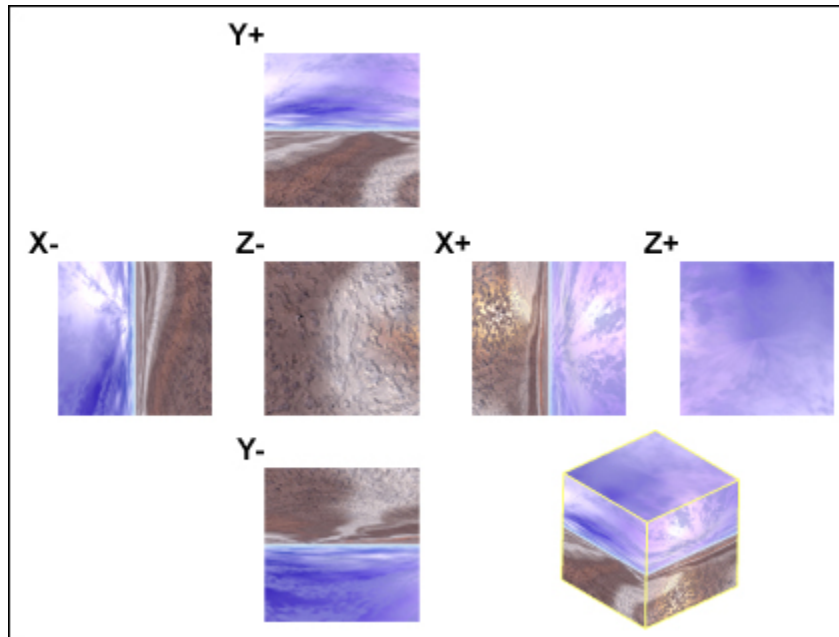
Static environment maps consist of pre-made 2D raster images that you manually load into the application. There are two types of static environment maps:

- Spherical — Spherical or standard environment maps wrap a single 2D image around the specified geometry.



An image used for the sphere environment map type

- Cube — Cube environment maps apply a different 2D image to each side of the specified geometry.



Images used for the cube environment map type

Dynamic cube environment mapping

Dynamic cube environment maps consist of automatically generated 2D images which capture the appearance of the geometry in the active 3D Viewing window.

Create static environment maps

Create environment maps consisting of pre-made 2D images. You can apply a single image to your parts or assemblies, or a different image for each of the six directional planes of the 3D scene.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Material** dialog box, choose the **Advanced** material type, and click **OK**.

A thumbnail image of the material appears on the Appearance Palette.

4. In the **Material** section, ensure the **Environment Map** check box is selected.

Tip:

Advanced materials are created with the **Color** and **Environment Map** options automatically selected. If you want to apply a reflection on top of an existing material without adding color, clear the **Color** check box.

5. (Optional) In the **Environment Map** section, from the **Layer** list, change the texture layer for the map.

Note:

By default, environment maps are created on texture **Layer 4**.

6. For **Type**, choose **Sphere** to use a single image as a reflection.

-or-

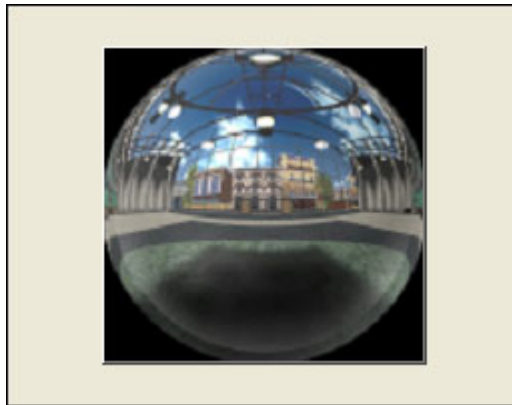
Choose **Cube** to use multiple images.

7. In the **Image** area of the **Environment Map** section, click the image box to bring up the Images page of the Appearance Editor, and then select an image.

-or-

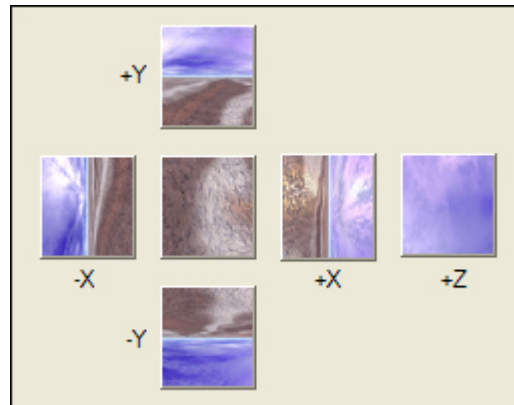
Load a new 2D image, and then click the image box.

On the Material page of the Appearance Editor, in the **Environment Map** section, a thumbnail image of the environment map appears.



An image used for the sphere environment map type

8. (If you chose **Cube**) Associate a 2D image for each of the remaining cube faces.



Images used for the cube environment map type

9. (Optional) If using a bump map with this environment map, select the **Use Clearcoat** check box to get the desired shiny effect.
10. In the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the material assignment.

11. Click **Set**.

In the Viewing window, the static environment map is assigned to the geometry.

Create dynamic environment maps

Create environment maps consisting of 2D images automatically generated from the model in the 3D Viewing window.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
2. On the Material page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Material** dialog box, choose the **Advanced** material type, and click **OK**.

A thumbnail image of the material appears on the Appearance Palette.

4. In the **Material** section, ensure the **Environment Map** check box is selected.

Tip:

Advanced materials are created with the **Color** and **Environment Map** options automatically selected. If you want to apply a reflection on top of an existing material without adding color, clear the **Color** check box.

5. (Optional) In the **Environment Map** section, from the **Layer** list, change the texture layer for the map.

Note:

By default, environment maps are created on texture **Layer 4**.

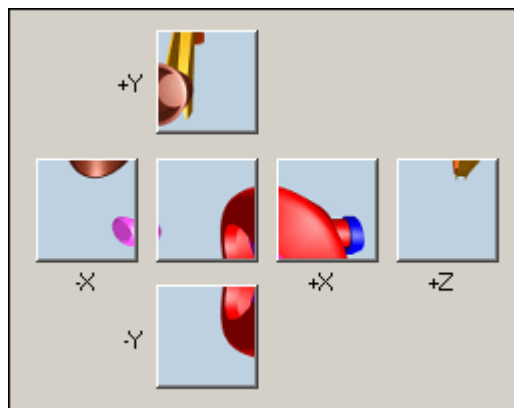
6. For **Type**, choose **Cube**.
7. (Optional) **Exclude parts** from the generated environment map images.

Note:

Unless you manually exclude parts from the creation of the environment map images, the parts to which you apply an environment map are excluded from the images. This helps prevent reflections from appearing on their parts of origin.

8. In the **Environment Capture** section, click **Capture**.

Six 2D images of your geometry are created, one for each of the six directional planes of the 3D scene (relative up, down, left, right, forward and backward) as seen from a **specified camera position**.



Six dynamically generated images of geometry

Note:

Parts must be visible to appear in environment map images.

9. (Optional) To automatically update the environment map images each time the 3D scene changes, select the **Capture when 3D change is detected** check box.

Note:

If this option is enabled, new images are generated whenever the appearance of the 3D parts in the Viewing window changes. This may be the result of a change to a material assigned to a part, one or more parts being shown or hidden, or part motion. You can prevent any of these actions from triggering updates with the **appearance preferences**.

10. (Optional) If using a bump map with this environment map, select the **Use Clearcoat** check box to get the desired shiny effect.
11. In the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the material assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the material assignment.

12. Click **Set**.

In the Viewing window, the dynamic environment map is assigned to the geometry.

Note:

If you do not manually **exclude parts** from the environment map images, the parts to which you assign the material are automatically excluded. However, after assigning the environment map to your parts, you may need to re-capture the images for the material to display as you intend.

Tip:

You can change the resolution used for the creation of environment map images in the **appearance preferences**.

Exclude parts from dynamic environment maps

Manually exclude parts from the images used to create dynamic environment maps.

1. **Create** or **edit** a dynamic environment map.

2. Select the parts you want to exclude.
3. On the Material page of the Appearance Editor, in the **Environment Capture** area of the **Environment Map** section, click **Parameters**.
4. In the **Environment Capture Parameters** section, for **Parts excluded from the environment capture**, click **Set from selected parts**.

The selected parts are not included in the images generated for the environment map.

Tip:

To identify excluded parts by selecting them in the 3D Viewing window, click **Show excluded parts**.

Change the dynamic environment map camera position

Change the position of the camera used to create the environment map images.

Example:

The rear view mirror in this model is a dynamic environment map set to update automatically as the 3D scene changes. The camera used to create the map images is positioned as if it is looking out from the center of the mirror, which results in realistic reflections that update as the camera perspective changes.



1. **Create** or **edit** a dynamic environment map.
2. On the Material page of the Appearance Editor, in the **Environment Capture** area of the **Environment Map** section, click **Parameters**.

3. In the **Camera Position** area of the **Environment Capture Parameters** section, do one of the following:
 - In the **Camera Position** boxes, type new **X**, **Y**, and **Z** coordinates.
 - Use the sliders to change the camera coordinates.
 - Click **Pick**, select the check box for one or more axes, and in the Viewing window, click a surface.
 - Click **Center**.

Note:

The camera position is initialized in the following ways:

- If the environment map is not assigned, the camera position is 0,0,0.
- After the environment map is assigned, the camera position is automatically changed to the center of the bounding box that encompasses all of the parts included in the material assignment.
- If the material assignment changes to include different parts, the camera position is not automatically updated to the new bounding box center. To manually update the camera position, you must click **Center**.

Update dynamic environment map images

You can manually update the images used for dynamic environment maps, or specify for the images to be automatically updated whenever the appearance of the 3D parts in the Viewing window changes.

Note:

Automatically generated images for dynamic environment maps are not saved with session files. Only the environment map parameters are saved. When you re-open the session file and turn on your parts, new images are automatically generated for the dynamic environment maps.

1. **Create** or **edit** a dynamic environment map.
2. Do any of the following:

To update the images	Do this
Manually, from the Appearance Editor	On the Materials page of the Appearance Editor, in the Environment Capture section, click Capture .
Manually, from the Appearance Palette	On the Materials page of the Appearance Palette, right-click the material containing the dynamic environment map, and select Capture Environment Images .
Automatically	On the Materials page of the Appearance Editor, in the Environment Capture section, select the Capture when 3D change is detected check box. Note: • New images are generated whenever the appearance of the 3D parts in the Viewing window changes due to a change to a material assigned to a part, one or more parts being shown or hidden, or part motion. You can prevent any of these actions from triggering updates with the appearance preferences. • The automatic image capture option creates all six of the images required for a dynamic environment map. You cannot combine pre-made 2D images with the images automatically generated by the application. Loading a 2D image into any of the environment map images boxes disables the Capture when 3D change is detected check box. Likewise, if you replace one of the automatically generated images with a different image, all of the images associated with the environment map are made static and the option is disabled.

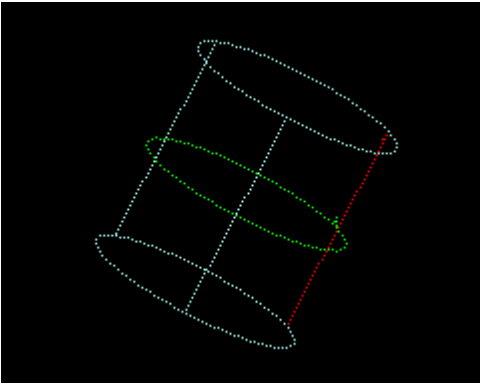
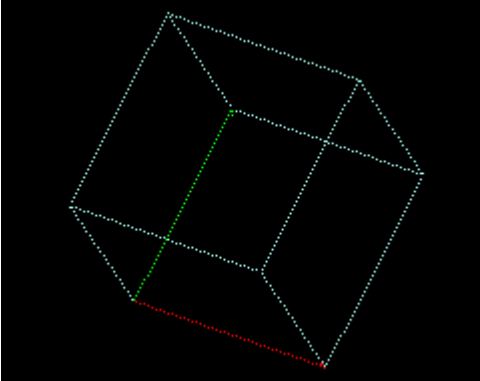
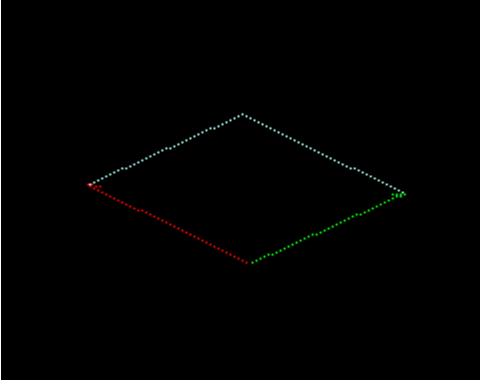
Working with texture coordinates

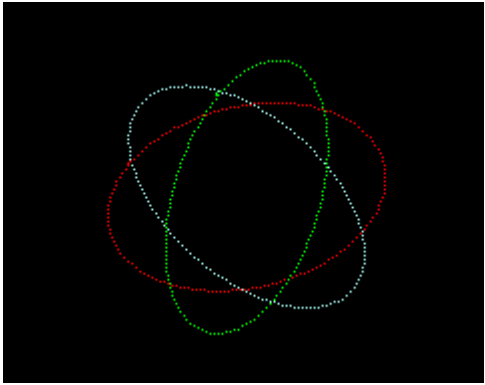
Overview of texture coordinate types

To map 2D images to 3D geometry, you first must apply texture coordinates to the surface of your geometry. You cannot apply a texture to a model that does not already have texture coordinates.

You can use the following types of texture coordinates:

Use this type of texture coordinate	For this
Cylindrical	The cylindrical mapping method enables you to evenly map a two-dimensional texture onto a three-dimensional cylindrical object. This mapping method is analogous to wrapping a label on a soda can. For example, if you assign a texture to a

Use this type of texture coordinate	For this
	cylinder using the cylindrical mapping method, the texture will be evenly mapped on the curving surface and distorted at the top and bottom of the projection.
Cubic 	The cube mapping method maps the texture image on every face of a cube.
Planar 	The planar mapping method maps a texture onto a model by collapsing the model geometry onto a plane and assigning texture coordinates to model vertices, based on the location of those vertices in the two-dimensional plane. This mapping method is analogous to projecting a movie image onto a wall. If the surface is not flat, distortion occurs.
Spherical	The spherical mapping method enables you to evenly map a two-dimensional texture onto a three-dimensional rounded surface.

Use this type of texture coordinate	For this
	

Manually create texture coordinates

For texture mapping to have a visual effect on geometry, the geometry also needs texture coordinates. When you create a texture map, bump map, or environment map, the texture coordinates are generated automatically. You can also manually create and manipulate texture coordinates.

1. On the **3D Appearance** toolbar, click **Edit Materials** .
2. On the Texture page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Texture Coordinates** dialog box, type a name for the texture coordinates.
4. Select a texture layer. Choose from 1 to 4.

Note:

You can create up to four layers of textures, one on top of another. To work with more than one texture, create texture coordinates for each layer.

5. Choose one of the following types of texture coordinates:
 - **Cylindrical**
 - **Cubic**
 - **Planar**
 - **Spherical**
6. Click **OK**.

A thumbnail image of the texture coordinates type appears on the Textures page of the **Appearance Palette**.

7. (Optional) In the **Texture Coordinates** section, select the **Show surface** check box.

The texture coordinates appear as a wireframe in the Viewing window.

8. (Optional) Adjust the **texture coordinates parameters** using the Appearance Editor.
9. (Optional) Adjust the texture coordinates parameters using **manipulators**.
10. **Assign the texture coordinates** to geometry in the Viewing window.

Tip:

Texture coordinates also can be assigned from the **Appearance Palette**.

Adjust texture coordinates in the Appearance Editor

You can use the Appearance Editor to change the size, position, and rotation values of texture coordinates.

Tip:

- You also can use **manipulators** to transform texture coordinates in the Viewing window.
- **Pick** options reference settings specified with the **3D selection filter**.

1. **Create** or edit texture coordinates.
2. Do any of the following:

For this	Do this
Resize the texture coordinates	In the Dimensions section, do any of the following: • Enter a value for each parameter. • Use the sliders. Note: The options available vary depending upon the texture coordinates type. For example, you can adjust spherical coordinates by horizontal and

For this	Do this
	vertical sweep, and radius, and you can adjust planar coordinates by width and height.
Reposition the texture coordinates	In the Origin section, do any of the following: • Enter a value for each parameter. • Use the sliders. • Click Pick, and in the Viewing window, click a surface. Note: The initial position of texture map coordinates is the center of the bounding box of the specified parts. Alignment is along the local coordinate axis for the parts. When specifying parts from different assemblies, alignment is along the world coordinate system.
Rotate the texture map coordinates	In the Rotation section, do any of the following: • Enter a value for each parameter. • Use the sliders. Note: To align an axis towards a picked point, click Pick, choose a rotation axis and an alignment axis, and then in the Viewing window, click a surface.

Manipulate texture coordinates

You can use manipulators to transform texture coordinates in the Viewing window. You can change the position, orientation, and size of texture coordinates.

1. **Create or edit texture coordinates.**
2. In the **Texture Coordinates** section, select **Show surface**.

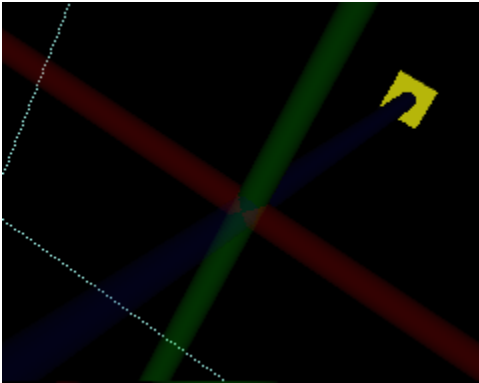
A wireframe representing the texture coordinates appears in the Viewing window.

3. Select **Enable 3D manipulators**.

The manipulators appear in the Viewing window, centered upon the texture coordinates.

4. Transform the texture coordinates in any of the following ways:

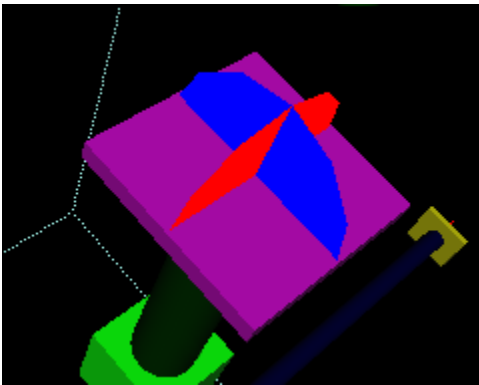
Translation manipulator



Translating texture coordinates changes the position of the coordinates relative to the geometry in the Viewing window.

There is a translation manipulator for each axis of the world coordinate system. To move texture coordinates along an axis, click and drag a translation manipulator.

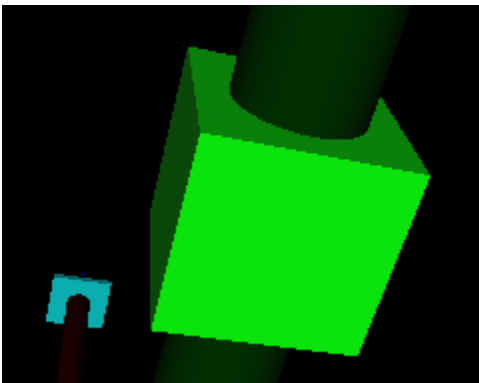
Rotation manipulator



Rotating texture coordinates changes the orientation of the coordinates relative to the geometry in the Viewing window.

To rotate texture coordinates, click and drag either of a rotation manipulator's intersecting half-circles.

Scaling manipulator



Scaling texture coordinates changes the size of the coordinates relative to the rest of the geometry in the Viewing window.

To scale texture coordinates, click and drag a scaling manipulator.

Adjusting texture parameters

Overview of texture filters

Texture maps are square or rectangular, but after being mapped to geometry and transformed, the individual texels (the individual values in a texture array) of a texture rarely correspond to individual

pixels of the final screen image. Depending upon the transformations used and the texture mapping applied, a single pixel on the screen can correspond to anything from a tiny portion of a texel (magnification) to a large collection of texels (minification). In both cases the question is how texel values should be used and how they should be averaged or interpolated. Consequently, several filtering options are provided to determine how texels correspond to screen pixels.

Because textured objects are viewed at different distances from the viewpoint, the texture map must be filtered down to an appropriate size without introducing visual artifacts. Artifacts can be avoided by using a series of prefiltered texture maps of decreasing resolutions, called mipmaps.

Mipmaps are automatically determined based upon the size (in pixels) of the object being mapped. With this approach, the level of detail in the texture map is appropriate for the image that is drawn on the screen. As the image of the object gets smaller, the size of the texture map decreases.

Specify texture filters

Change the texture filter to control the texture image quality.

1. For **Min**, choose one of the following:

Minification texture filter	Filter description	Effect on image and performance
Nearest	Uses the nearest texel value the original image	Requires the least computation time of the filters. Recommended for low-end systems, but may result in severe aliasing artifacts.
Linear	Uses a weighted linear average of the 2x2 array of texels that are nearest to the center of the pixel.	Requires more computation than Nearest filter. Provides smoother results.
Nearest In Mip	Uses the nearest texel value in the MIPMAP image resolution determined by the pixel size. Think of the MIPMAP as a pyramid, with the original image at the bottom, and increasingly smaller images towards the top. Nearest In Mip chooses one of the smaller images, and takes the nearest texel.	Faster but yields less desirable results.
Linear In Mip	Interpolates linearly. Think of the MIPMAP as a pyramid, with the original image at the bottom, and increasingly smaller images towards the top. Linear In Mip chooses one of the smaller images, and then interpolates linearly between 4 texels.	Requires more computation than Nearest in Mip filter but provides better results.

Minification texture filter	Filter description	Effect on image and performance
Bilinear Mip	Select the nearest texel in each of the two texture maps and then interpolates linearly between these two values.	Selecting between two maps requires more computation but provides better results.
Trilinear Mip	Uses linear interpolation to compute the value in each of two texture maps and then interpolates linearly between these two values.	Produces the smoothest results but requires the most computation.

2. For **Mag**, choose one of the following:

Magnification texture filter	Filter description	Effect on image and performance
Nearest	Uses the texel with coordinates nearest the center of the pixel.	Requires less computation than Linear filter. Can result in severe aliasing artifacts.
Linear	Uses a weighted linear average of the 2x2 array of texels that are nearest to the center of the pixel.	Requires more computation than Nearest filter. Provides smoother results.

Choose a texture blend color

- For **Blend Color**, click the **color well**, and choose a color.

Define a texture border

- Select the **Implicit Border** check box.
- For **Color**, click the **color well**, and choose a color.
- For **Transparency**, type a value between 0 and 1.

Note:

- When mapping a texture in Clamp mode, the border pixels of the texture are repeated (smeared) over the remaining geometry. To avoid this, you can use an image that has a uniform row of border texels in the required color. **Implicit Border** allows you to define the color of a uniform border, which is then added automatically.
- A border with a transparency of 1 is invisible; a border with a transparency of 0 is opaque.
- You can also use the slider.

Adjust texture placement options

Use the texture placement options to adjust the size, placement, and rotation values for texture maps, bump maps, and environment maps.

1. Create or **edit** a **texture map**, **bump map**, or **environment map**.
2. In the **Texture Map**, **Bump Map**, or **Environment Map** section, click **Placement**.

The appropriate **Image Placement** section appears.

3. Do any of the following:

To	Do this
Resize the image	a. Select Scale. b. For S (horizontal), do one of the following: • Enter a value. • Use the slider. c. For T (vertical), do one of the following: • Enter a value. • Use the slider.
Reposition the image	a. Select Translation. b. For S (horizontal), do one of the following: • Enter a value. • Use the slider. c. For T (vertical), do one of the following: • Enter a value. • Use the slider.
Rotate the image	a. Select Rotation. b. For R (rotation), do one of the following:

To	Do this
	• Enter a value. • Use the slider.

4. (For texture maps and bump maps only) For **Wrapping**, choose one of the following wrapping methods for the **S** (horizontal) and **T** (vertical) values:

This wrapping method	Does this
Clamp	A single copy of the image appears on the material and the remainder of the surface (not covered by the texture) is applied with the texture border colors.
Repeat	The image is duplicated across the material surface.
Mirror repeat	The image is duplicated across the material surface, with every other instance of the image inverted for a mirrored effect.

Blending textures

You can use different blending methods to determine how the color values of a texture combine with other textures or with the underlying color values of the material.

The following texture blending methods are available:

This method	Does this
Decal	The texture image is applied as if the image were a decal placed over the material. As a result, the shading of the underlying material is not blended with the texture image. Note: For textures with an alpha channel, Decal mixes the image colors with the material color according to the alpha channel.
Modulate	The texture image modulates or scales the surface color. This is useful for combining the effects of lighting with texturing. Note: Use the Modulate blending method to create a texture that responds to lighting conditions.
Replace	The texture image completely replaces the model's material color.

This method	Does this
	Note: For textures with or without an alpha channel, Replace results in the image completely covering the underlying material color.
Blend	The texture image blends with the underlying material and lets the shading of the model be seen. When using blending, the texture serves as a filter for the underlying material color. Each of the red, green, blue, and alpha values is determined by averaging the values of the texture and material pixels. Note: For textures created using Blend, you must specify the blend color.
Add	The RGB values for the texture image and the underlying material are combined.

Assigning and manipulating materials

Assign materials to geometry from the Appearance Editor

To see part appearance changes in the Viewing window, you must assign your materials to geometry.

Tip:

You also can assign materials from the **Appearance Palette**.

1. On the Material page of the Appearance Editor, in the **Assignments** section, do any of the following:

To	Do this
Assign the material to all visible parts, but not override any part shape assignments	Choose Visible parts , clear the Specific occurrence only check box, and then click Set .
Assign the material to all visible parts, and suppress previously assigned materials and part shape assignments	Choose Visible parts , select the Specific occurrence only check box, and then click Set .
Assign the material to selected parts, but not override any part shape assignments	Select the parts, choose Selected parts , clear the Specific occurrence only check box, and then click Set .

To	Do this
Assign the material to selected parts, and suppress previously assigned materials and part shape assignments	Select the parts, choose Selected parts , select the Specific occurrence only check box, and then click Set .
Assign the material to selected assemblies, but only change the appearance of parts with no previously assigned materials	Select the assemblies, choose Selected assemblies or parts , clear the Specific occurrence only check box, and then click Set .
Assign the material to selected assemblies, and change the appearance of all parts in the assemblies (Suppress materials previously assigned to parts in the assemblies)	Select the assemblies, choose Selected assemblies or parts , select the Specific occurrence only check box, and then click Set .
(For assemblies with multiple instances of a part) Assign the material to a selected instance of the part	Select the part instance, choose Selected parts , select the Specified occurrence only check box, and then click Set .
(For assemblies with multiple instances of a part) Assign the material to all instances of the part	Select one part instance, choose Selected parts , clear the Specified occurrence only check box, and then click Set .
Assign materials to faces Note: You must have V9.1 or later JT data to assign materials to faces. With pre-V9.1 JT files, picking a face picks the parent shape.	- Choose Sub-part geometry. - In the Viewing window click a face that you want to include in the material assignment. You can hold Ctrl to select more than one face, or hold Shift to select all the face groups with the same material within the same part. Note: If the selected face groups represent an entire shape, a shape assignment will be made. - Click Set.
Assign materials to shapes	- Choose Sub-part geometry. - Double-click a shape and click Set.

To	Do this
	Note: If you select a shape that has only one face, the material assignment will be made to the face.
Identify all parts in the Viewing window with an assigned material	- Choose Target and then choose Selected assemblies or parts from the dropdown. - Click Select.
Identify all shapes or faces in the Viewing window with an assigned material.	- Choose Sub-part geometry. - Click Select.
Remove an assigned material	Click Remove .
Undo the last assignment action	Press Ctrl+Z. Note: To reverse an undo, press Ctrl+Y.

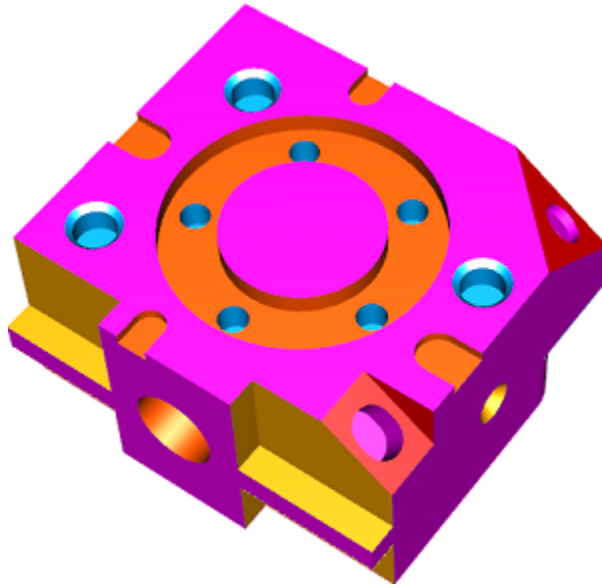
Overview of sub-part geometry and materials

Parts are made up of one or more shapes. Shapes are made up of faces.

Shapes and faces are both considered *sub-part geometry*. You can assign materials to both types of sub-part geometry.

Note:

You must have V9.1 or later JT data to assign materials to faces. With pre-V9.1 JT files, picking a face picks the parent shape.



Overview of assembly-level material assignments

Materials assigned to shapes, parts, and assemblies affect the appearance of geometry in the Viewing window.

In the Appearance Editor, you can create, edit, and assign materials. When you assign materials to assemblies, you can choose how the materials affect the appearance of parts within the assembly. Materials assigned to assemblies can:

- Change the appearance of the entire assembly. Assembly-level material assignment can suppress materials assigned to individual parts and subassemblies within the assembly.
- Change the appearance of only those assembly parts with no materials assigned. Materials assigned to individual parts continue to affect their appearance.

When you open a model with materials assigned to assemblies, you see the effects of the assembly-level material assignments on the parts. When you open an individual part of the same model, you see the effect of material assigned to the individual part.

Edit materials

Edit materials in the Appearance Editor.

- Do any of the following to load a material into the Appearance Editor:

To edit materials	Do this
From the Appearance Palette	Do one of the following: • Double-click a Palette entry. • Select a Palette entry, and click Edit. • Right-click a Palette entry, and choose Edit.
From the Appearance Editor	1. On the 3D Appearance toolbar, click Edit Materials . 2. In the Management section, choose one of the following: • Visible parts • Selected assemblies or parts (materials and lights only) • All models (lights only) 3. Do one of the following: • (If you chose Visible parts) Display the part with the material or light that you want to edit, and click Retrieve. • (If you chose Selected assemblies or parts) Select a part with the material or light that you want to edit, and click Retrieve. • (If you chose All models) Click Retrieve.

The material settings appear in the Appearance Editor. You can adjust the settings as you like.

Preview materials

You can preview your material within the Appearance Editor. As you adjust material settings, the preview image updates to reflect your changes.

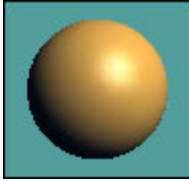
Tip:

If you assign your material to geometry while adjusting its settings, you can see your changes in the Viewing window as you make them.

1. On the **3D Appearance** toolbar, click **Edit Materials** .

- On the Material page of the Appearance Editor, expand the **Preview** section.

A sphere displays the current material settings as they appear when assigned to geometry in the Viewing window.



Copy materials

You can copy existing materials.

- On the **3D Appearance** toolbar, click **Edit Materials** .

The Material page of the Appearance Editor appears.

- In the **Management** section, select **Update when 3D selection or visibility changes**.
- Specify to copy the material from **Visible parts** or **Selected assemblies or parts**.
- (If you chose **Visible parts**) Display the part with the material that you want to copy.

-or-

(If you chose **Selected assemblies or parts**) Select an assembly or part with the material that you want to copy.

- Click **Retrieve**.

The material settings appear in the Appearance Editor. You can adjust the settings as you like.

Remove materials from parts

You can remove materials from parts and assemblies

- On the **3D Appearance** toolbar, click **Edit Materials** .

The Material page of the Appearance Editor appears.

- In the **Management** section, choose **Selected assemblies or parts**.
- Select **Update when 3D visibility or selection changes**.

4. In the Viewing window, select the part from which you want to remove a material.

The material's settings appear in the Appearance Editor.

5. In the **Assignments** section, click **Remove**.

The material is removed from the part.

Assigning and manipulating materials

Use the Appearance Editor to create, edit, and to assign materials to parts in the Viewing window. You also can remove materials from parts, and restore parts to the way they looked when first loaded.

Restore the original appearance of parts

If you assign materials to parts and assemblies, you can restore them to their original appearance when first loaded.

1. On the **3D Appearance** toolbar, click **Edit Materials** .

The Material page of the Appearance Editor appears.

2. In the **Management** section, choose **Selected assemblies or parts**.
3. Select **Update when 3D visibility or selection changes**.

4. In the Viewing window, select the part that you want to restore to its original appearance when first loaded.

The material's settings appear in the Appearance Editor.

5. (If you are restoring an assembly that contains sub-assemblies) In the **Assignments** section, select **Restore all sub-assemblies**.
6. Click **Restore**.

The selected assemblies or parts are restored or their original appearance when first loaded.

Note:

You cannot restore the appearance of single **shapes** without restoring the appearance of the entire part.

Assign texture coordinates from the Appearance Editor

You can manually assign texture coordinates to your geometry for greater control over the position and appearance of **texture maps** and **bump maps**.

1. **Create** or **edit** texture coordinates.
2. In the **Assignments** section, do one of the following:
 - Choose **Visible parts**, and display only parts that you want to include in the texture coordinates assignment.
 - Choose **Selected parts**, and select parts that you want to include in the texture coordinates assignment.
3. Click **Set**.

The texture coordinates are assigned to the geometry in the Viewing window.

Tip:

- Click **Select** to identify all parts in the Viewing window with the assigned texture coordinates.
- Texture coordinates also can be assigned from the **Appearance Palette**.

Note:

- Use **Add** when working with multiple texture layers. You can create up to four separate layers of texture coordinates.
- Click **Remove** to remove the assigned texture coordinates.

Saving materials with models

When you save your work as session files, changes to materials and material assignments are saved.

When you export models to the JT format, materials and material assignments are saved in the JT files.

When you save or export to the PLM XML format, you can choose to include materials and material assignments.

Note:

You can assign materials to faces, shapes, parts, and assemblies. When you save or export models with assembly-level material assignments, results depend on the type of file opened, where the assignments were authored, and the output file format.

If you open	Then export	The result is
PLM XML or JT files, and then assign materials to assemblies	PLM XML or JT files	Assembly-level material assignments are preserved.
PLM XML files with assembly-level material assignments not authored in Teamcenter lifecycle visualization	JT files	Assembly-level material assignments are preserved.
PLM XML files with assembly-level material assignments not authored in Teamcenter lifecycle visualization	PLM XML files	Assembly-level material assignments are preserved.
JT files with assembly level-material assignments not authored in Teamcenter lifecycle visualization	JT files	Assembly-level material assignments are preserved.
JT files with assembly level-material assignments not authored in PLM XML	PLM XML files	Assembly-level material assignments are lost.

Working with lights

Lights overview

You can create different types of light sources to illuminate geometry in the 3D Viewing window. Each light type generates unique lighting effects. You also can configure light sources to cast shadows, specify default lighting, and adjust various light settings.

Create directional lights

Directional lights have no position, only direction. Create them when you want to simulate infinite light sources like the sun.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Light** dialog box, from the **Type** list, choose **Directional**, and then click **OK**.

A thumbnail image of the light appears on the Appearance Palette.

Note:

By default, newly created lights are automatically assigned to the model root. You can change this behavior in the **Appearance Preferences**.

4. In the **Light** section, for **Coordinate System**, choose **World** if you want to **display the light source** in the Viewing window.

-or-

Choose **View** if you want the light source to move with the camera and not be visible in the Viewing window.

5. (Optional) Turn on **shadows**.
6. (Optional) Change the light **color and intensity**.
7. (Optional) In the **Position** section, change the light position in any of the following ways:
 - In the **Position** boxes, type new X, Y, and Z coordinates.
 - Use the sliders to change the light coordinates.
 - Click **Pick**, select the check box for one or more axes, and in the Viewing window, click a surface.
8. In the **Direction** section, choose **Direction vector** if you want to change the light direction using a specified direction vector.

-or-

Choose **Target point** if you want to change the light direction using a specified target point on the model.

9. Change the light direction in any of the following ways:
 - In the **Direction** boxes, type new i, j, and k coordinates.
 - Use the sliders.
 - Click **Pick**, select the check box for one or more axes, and in the Viewing window, click a surface.
 - Use the **light manipulators**.
 - Use the **Direction Manipulator**.

10. In the **Assignments** section, do one of the following:

To	Do this
Assign the light to visible parts	Choose Visible parts , and display only the parts that you want to include in the light assignment.
Assign the light to selected parts or assemblies	Choose Selected assemblies or parts , and select parts or assemblies that you want to include in the light assignment.
Assign the light to all open models	Choose Models .

11. Click **Set**.

In the Viewing window, the light is assigned to the specified geometry.

Create point lights

Point lights are omni-directional. Create them when you want to simulate indoor light sources.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Light** dialog box, from the **Type** list, choose **Point**, and then click **OK**.

A thumbnail image of the light appears on the Appearance Palette.

Note:

By default, newly created lights are automatically assigned to the model root. You can change this behavior in the **Appearance Preferences**.

4. In the **Light** section, for **Coordinate System**, choose **World** if you want to **display the light source** in the Viewing window.

-or-

Choose **View** if you want the light source to move with the camera and not be visible in the Viewing window.

5. (Optional) Turn on **shadows**.
6. (Optional) Change the light **color and intensity**.

7. (Optional) In the **Position** section, change the light position in any of the following ways:
 - In the **Position** boxes, type new X, Y, and Z coordinates.
 - Use the sliders to change the light coordinates.
 - Click Pick, select the check box for one or more axes, and in the Viewing window, click a surface.
8. In the **Assignments** section, do one of the following:

To	Do this
Assign the light to visible parts	Choose Visible parts , and display only the parts that you want to include in the light assignment.
Assign the light to selected parts or assemblies	Choose Selected assemblies or parts , and select parts or assemblies that you want to include in the light assignment.
Assign the light to all open models	Choose Models .

9. Click **Set**.

In the Viewing window, the light is assigned to the geometry.

Create spot lights

Spot lights have both position and direction. Create them when you want to light a particular area within a scene.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Light** dialog box, from the **Type** list, choose **Spot**, and then click **OK**.

A thumbnail image of the light appears on the Appearance Palette.

Note:

By default, newly created lights are automatically assigned to the model root. You can change this behavior in the **Appearance Preferences**.

4. In the **Light** section, for **Coordinate System**, choose **World** if you want to **display the light source** in the Viewing window.

-or-

Choose **View** if you want the light source to move with the camera and not be visible in the Viewing window.

5. (Optional) Turn on **shadows**.
6. (Optional) Change the light **color and intensity**.
7. In the **Spot Light Attributes** section, in the **Cut-off Angle** box, type a value between 0 and 90 to control the width of the spot light.

-or-

Adjust the light width with the sliders.

8. In the **Spot Exponent** box, type a value between 0 and 100 to control the light attenuation from the center axis of the light source towards the edges of the cone.

-or-

Adjust the attenuation with the sliders.

9. (Optional) In the **Position** section, change the light position in any of the following ways:
 - In the **Position** boxes, type new X, Y, and Z coordinates.
 - Use the sliders to change the light coordinates.
 - Click **Pick**, select the check box for one or more axes, and in the Viewing window, click a surface.
10. In the **Direction** section, choose **Direction vector** if you want to change the light direction using a specified direction vector.

-or-

Choose **Target point** if you want to change the light direction using a specified target point on the model.

11. Change the light direction in any of the following ways:
 - In the **Direction** boxes, type new i, j, and k coordinates.
 - Use the sliders.
 - Click **Pick**, select the check box for one or more axes, and in the Viewing window, click a surface.

- Use the **light manipulators**.
- Use the **Direction Manipulator**.

12. In the **Assignments** section, do one of the following:

To	Do this
Assign the light to visible parts	Choose Visible parts , and display only the parts that you want to include in the light assignment.
Assign the light to selected parts or assemblies	Choose Selected assemblies or parts , and select parts or assemblies that you want to include in the light assignment.
Assign the light to all open models	Choose Models .

13. Click **Set**.

In the Viewing window, the light is assigned to the geometry.

Overview of light maps

Light maps use textures containing pre-computed lighting information to give 3D models a more realistic appearance. When a light map is assigned to geometry, the amount of light displayed for any given vertex is taken directly from the texture, which can use any light map types, such as High Dynamic Range (HDR) or other raster formats such as TIFF or JPEG. Lighting effects that make use of greater color precision produce 3D scenes with fewer color banding chromatic artifacts. You can use light maps to illuminate parts and assemblies.

Latitude/Longitude texture mapping is used to generate light maps in the application. Similar to spherical texture maps, Latitude/Longitude textures directly map a 2D image onto a hollow sphere, and then use the reflected view vector to map the sphere onto geometry.

You may create light maps using **OpenEXR** or Radiance RGBE Encoding images. A Photoshop OpenEXR plug-in is available on the OpenEXR website at <http://www.openexr.com/>.

Note:

Light maps are supported only on hardware that supports programmable shaders. This includes the NVIDIA Quadro FX 500 or later graphics cards, and the ATI FireGL T2, X3, and V3100 and later graphics cards. For light maps to display correctly, make sure you have the latest drivers for your card.

Light maps are supported only when using GL2.1 or higher.

Create light maps

Light maps use High Dynamic Range (HDR) or other raster images to simulate global illumination. Create them when you want to give your 3D scenes more realistic lighting effects.

Note:

- Light maps cannot cast shadows because they do not have a location or direction.
- Light maps always use the World coordinate system.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Management** section, click **New**.
3. In the **New Light** dialog box, from the **Type** list, choose **Map**, and then click **OK**.

A thumbnail image of the light appears on the Appearance Palette.

Note:

By default, newly created lights are automatically assigned to the model root. You can change this behavior in the [Appearance Preferences](#).

4. On the Light page of the Appearance Editor, in the **Image** section, click the image box to bring up the Images page of the Appearance Palette, and then select a raster image.

-or-

Load a raster image, and then click the image box.

A thumbnail image of the light map appears.

5. In the **Orientation** section, change the orientation of the light map in any of the following ways:
 - In the **X**, **Y**, or **Z** boxes, type new coordinates.
 - Use the sliders.

Note:

Orientation values are in degrees.

6. In the **Intensity** box in the **Parameters** section, enter a value to define the intensity of the light map.

7. In the **Assignments** section, do one of the following:

To	Do this
Assign the light to visible parts	Choose Visible parts , and display only the parts that you want to include in the light assignment.
Assign the light to selected parts or assemblies	Choose Selected assemblies or parts , and select parts or assemblies that you want to include in the light assignment.
Assign the light to all open models	Choose Models .

8. Click **Set**.

In the Viewing window, the light is assigned to the geometry.

Assign lights

You must assign your lights to geometry to see lighting effects in the Viewing window.

1. **Create** or **edit** a light.
2. On the Lights page of the Appearance Editor, in the **Assignments** section, choose **Visible parts**, and display only the parts that you want to include in the light assignment.

-or-

Choose **Selected assemblies or parts**, and select parts or assemblies that you want to include in the light assignment.

-or-

Choose **Models**.

3. Click **Set**.

In the Viewing window, the light is assigned to the geometry.

Tip:

- Click **Select** to identify all parts in the Viewing window with the light assignment.
- You also can assign lights from the **Appearance Palette**.

Note:

- Click **Add** to assign the light more than once. Lights assigned multiple times to the same geometry have a cumulative effect.
- Click **Remove** to remove the light from the part's light assignments.
- If a light is assigned to an assembly or subassembly, it affects all of its subassemblies and parts.

Edit lights

You can edit the settings of existing light sources.

1. Do any of the following:

To access light settings	Do this
On the Lights page of the Appearance Palette	Double-click a light icon. -or- Select a Palette entry, and click Edit . -or- Right-click a light icon, and select Edit .
On the Light page of the Appearance Editor	a. On the 3D Appearance toolbar, click Edit Lights . b. On the Light page of the Appearance Editor, in the Management section, choose Visible parts, and display only the parts with the light that you want to edit. -or- Choose Selected assemblies or parts, and select a part with the light that you want to edit. -or- Choose Models. c. Click Retrieve.

The settings for the light source appear in the Appearance Editor.

2. Update the light settings.

Copy the settings of existing lights

You can create new light sources quickly by copying the settings from existing lights.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Management** section, select **Update when 3D selection or visibility changes**.
3. Choose **Visible parts**, and display only parts with the light assignment that you want to copy.

-or-

Choose **Selected assemblies or parts**, and **select** an assembly or part with the light assignment that you want to copy.

-or-

Choose **Models** to copy the settings from a light assigned to the top-level assembly node of the model.

4. Click **Retrieve**

The light settings appear in the Appearance Editor.

5. Click **Clone**.

A new light entry appears on the Lights page of the Appearance Palette.

6. Update the light settings.
7. **Assign the light** to geometry in the Viewing window.

Define default lighting

The default light source is automatically applied to models when you open them.

Tip:

Generally, a light created using the View Coordinate System is most useful as a default light.

1. On the **3D Appearance** toolbar, click **Light Palette** .

2. On the Light page of Appearance Palette, select one or more light entries.
3. Right-click the selected light entries, and choose **Set As Default Lights**.

When you open a 3D document, the default light settings are automatically applied.

Change light color and intensity

You can change both the color and intensity of directional lights.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Color** section, click the **color well** to change the light color.
3. For directional lights, drag the intensity slider left or right to adjust light brightness.

Tip:

Type a higher value in the box to the right of the slider for finer control of light brightness.

Turn shadows on or off

You can turn shadows on and off for directional lights.

1. In the Viewing window, right-click and choose **Performance**→**Edit**.
2. On the **Rendering** page of the **Performance** dialog box, select the **Cast shadows defined by light sources** check box.

Note:

- You can choose to turn on shadows only when geometry or the camera is not moving (still frames), only when geometry or the camera is moving (moving frames), or both.
- You can adjust the softness of shadow edges and the resolution of shadow maps on the **Global** page of the Performance dialog box.

3. On the **3D Appearance** toolbar, click **Edit Lights** .
4. On the Light page of the Appearance Editor, in the **Shadows** section, select the **Cast Shadows** check box.
5. Type a value between zero and one in the **Opacity** box to adjust the shadow opacity.

-or-

Adjust the slider between **Light** and **Dark**.

Note:

An opacity value of zero makes shadows appear solid; a value of one makes them appear completely opaque. The default value is .5.

Turn light sources on or off

You can turn light sources on or off using the Appearance Editor or the Appearance Palette.

Do either of the following:

To turn light sources on or off from	Do this
The Appearance Editor	- On the 3D Appearance toolbar, click Edit Lights . - On the Light page of the Appearance Editor, in the Light section, select or clear the Light on check box.
The Appearance Palette	- On the 3D Appearance toolbar, click Light Palette . - On the Lights page of Appearance Palette, right-click a light icon, and select or clear Light On.

Display light sources

You can specify to display light sources in the Viewing window using the Appearance Editor or the Appearance Palette.

Do either of the following:

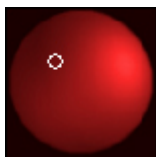
To display light sources from	Do this
The Appearance Editor	- On the 3D Appearance toolbar, click Edit Lights . - On the Light page of the Appearance Editor, in the Light section, select or clear the Show light check box.
The Appearance Palette	On the 3D Appearance toolbar, click Light Palette .

To display light sources from	Do this
	2. On the Lights page of Appearance Palette, right-click a light icon, and select or clear Show Light .

Use the Direction Manipulator

Use the Direction Manipulator to control the direction of directional lights.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Light** section, select **View** from the **Coordinate System** list.
3. In the **Direction Manipulator** section, select the direction vector. Choose from
 - -Z
 - +Z
4. Click anywhere in the Direction Manipulator and drag the small white circle to adjust the direction of the light source.



The Direction Manipulator

Manipulate lights

Use the light manipulator to change the position of directional lights in the Viewing window.

1. On the **3D Appearance** toolbar, click **Edit Lights** .
2. On the Light page of the Appearance Editor, in the **Light** section, select the **Show light** check box.

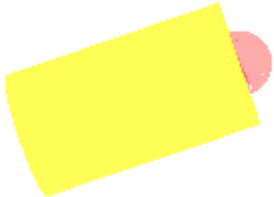
Tip:

You may need to adjust the camera or turn off geometry to see the light source. You also may need to switch the light source to the World coordinate system.

3. Select the **Enable 3D manipulator** check box.

The manipulator appears on the light source.

4. Manipulate the light in any of the following ways:

To	Do this
Change the light position	In the Viewing window, click and drag the light to a new position.
Change the light direction	In the Viewing window, click and drag the base of the light to adjust its direction.  A directional light
Aim the light source at a target	a. In the Direction section, choose Target point. b. Click Pick, and then, in the Viewing window, click a location on your model. The light source is aimed at your target.

Working with the appearance palette

Overview of the Appearance Palette

Use the Appearance Palette to manage materials, 2D images loaded for use as textures, texture coordinates, and lights.

Appearance Palettes are associated with particular documents. You can switch between the Appearance Palettes of open documents or the Appearance Palettes of **PLM XML file appearance libraries** that you have loaded.

You also can work with the Current Document palette, and the Working Palette. The Current Document Palette is intended for use when you are working with multiple documents in the Viewer, displaying the Appearance Palette entries for the currently active document. The Working Palette functions as a

temporary location where you can store Appearance Palette entries while you continue to work within the Appearance Editor.

Assign appearance attributes from the Appearance Palette

Use the Appearance Palette to assign materials and lights to geometry in the Viewing window.

Tip:

You also can assign materials from the **Appearance Editor**.

1. On the **3D Appearance** toolbar, click one of the following:

- **Material Palette** 

- **Light Palette** 

The appropriate page of Appearance Palette appears.

Tip:

You can enable more than one Appearance Palette at a time.

2. (Optional) If working with multiple files or the Working Palette, in the upper left corner of the page, select one of the following:

- **Current Document**
- **[the name of an open file]**
- **Working Palette**

The Palette contents appear.

3. Do any of the following:

To	Do this
Assign a Palette entry	a. Select an Appearance Palette entry. The Appearance Palette entry is highlighted with a green outline. b. Specify the scope of the assignment by choosing one of the following:

To	Do this
	• Visible parts • Selected assemblies or parts • All models (lights only) c. (If you chose Visible parts) Display only parts that you want to include in the assignment. d. (If you chose Selected assemblies or parts) Select only assemblies or parts that you want to include in the assignment. e. Click Assign To. The Appearance Palette entry is assigned to the contents of the Viewing window as you specified. Note: • Assignments made from the Appearance Palette are always for the specified part occurrence only. • If you assign a Materials attribute from a loaded palette, a copy of the assigned attribute is placed onto the Current Document palette. You can make changes to, or rename, the copied attribute on the Current Document palette without affecting the original attribute on the loaded palette. • You can use the Undo command to undo the last Assign action. Tip: To identify assigned parts in the Viewing window, right-click an Appearance Palette entry, and choose Select Parts.
Preview a Palette entry	a. Click Try On. b. Specify the scope of the preview by choosing one of the following: • Visible parts • Selected assemblies or parts

To	Do this
	• All models (lights only) c. (If you chose Visible parts) Display only parts that you want to include in the preview. d. (If you chose Selected assemblies or parts or Selected parts) Select only assemblies or parts that you want to include in the preview. e. Select an Appearance Palette entry. The Appearance Palette entry is highlighted with a green outline, and assigned to the contents of the Viewing window as you specified. Tip: Try On lets you quickly apply different Appearance Palette entries to geometry in the Viewing window. When you click Try On, select different Appearance Palette entries to see their effects on geometry in the Viewing window. Click Stop to keep an assignment.
Edit a Palette entry	Do any of the following: • Select a palette entry, and click Edit. • Double-click a palette entry. • Right-click a palette entry, and select Edit. The values of the Appearance Palette entry appear in the Appearance Editor.
Send a material from the Appearance Editor to Working Palette	On the Material page of the Appearance Editor, in the Preview section, click Save .
Swap a material in the Appearance Editor with a material on the Working Palette	a. Select a material entry on the Working Palette. b. On the Material page of the Appearance Editor, in the Preview section, click Swap.
Switch between open Palettes	In the upper left corner of the Appearance Palette, choose a library.
Close a palette	Right-click the Palette, and choose Unload Palettes .

To	Do this
Create a new Palette entry	Right-click the Palette, and choose New .
Create a copy of a Palette entry	Right-click a Palette entry, and select Clone .
Delete a Palette entry	Right-click a Palette entry, and select Delete. Tip: You can use Ctrl and Shift to select multiple Palette entries. Note: You can use the Undo command to undo the last Delete action.
Rename a Palette entry	Right-click a Palette entry, and select Rename .
Copy a Palette entry from one document Palette to another	a. Right-click a Palette entry, and select Copy. Tip: You can use Ctrl and Shift to select multiple Palette entries. b. In the upper left corner of the page, choose another document Palette. The Palette that you want to copy to appears. c. Right-click the Palette, and select Paste. Tip: You can open multiple Appearance Palette dialog boxes at a time, and copy/paste from one to another.
Copy the complete contents of a document Palette to another	a. Right-click a Palette, and choose Select All. Each Palette entry is highlighted with a red outline. b. Right-click the Palette, and select Copy. c. In the upper left corner of the page, choose another document Palette.

To	Do this
	The Palette that you want to copy to appears. d. Right-click the Palette, and select Paste .

Load images for use with textures

You can load 2D raster images for use as textures.

1. On the **3D Appearance** toolbar, click **Image Palette** .
2. On the Images page of the Appearance Palette, right-click an empty area and choose **New**.
3. In the **New Image** dialog box, browse to the image location, and load your file.

The image appears on the Appearance Palette.

Filter Appearance Palette entries

CAD systems can arbitrarily assign colors to parts. If you load a model that consists of many parts, you may see a large number of unnamed materials on the Appearance Palette. You can control what content is displayed on the Palette, including images and lights, by choosing to show or hide named, unnamed, or unassigned Palette entries.

- On the **3D Appearance** toolbar, click one of the following:
 - **Material Palette** 
 - **Light Palette** 

The appropriate page of Appearance Palette appears.

Tip:
You can enable more than one Appearance Palette at a time.

- Right-click an empty area on the Appearance Palette, and choose any combination of the following filter options:

Choose this option	To do this
Show Names	Show the names of Appearance Palette entries.
Show Unnamed	Show Appearance Palette entries without names.
Show Unassigned	Show Appearance Palette entries that have not been assigned to geometry.

Reuse appearance palettes with other models

You can save materials and lights from one session and reuse them with other models in other sessions.

From the Appearance Palette window, you can save appearance palettes to an appearance library PLM XML file. You can then load or merge saved appearance palettes into the Appearance Palette for other sessions.

1. If the Appearance Palette is not already open, on the **3D Appearance** toolbar, click one of the following:

- **Material Palette** 

- **Light Palette** 

The appropriate page of Appearance Palette appears.

Tip:

You can enable more than one Appearance Palette at a time.

2. Right-click an empty area on the Appearance Palette, and do any of the following:

To	Do this
Save palettes to an appearance library PLM XML file	a. Choose Save Palettes. b. In the Save Palettes dialog box, select the palettes to include in the appearance library file. • Materials • Images • Texture Coordinates • Lights

To	Do this
	c. Click Browse to specify the location and file name. d. To save copies of images, select Copy all images to directory and specify a directory. Note: Image palettes may include both images embedded in the JT and references to external image files. • If you choose this option, all images are saved as .tif files to the specified directory. • If the palettes includes references to external image files, and you do not choose this option, the saved palettes will include the original references. • If the palettes include images embedded in the JT, and you do not choose this option, the Image Save Warning dialog box will appear so you can save the image files.
Load palettes	a. Choose Load Palettes. b. In the Load Palettes dialog box, select what to load. Choose from • Materials • Images • Texture Coordinates • Lights c. Click Browse, navigate to the location of the appearance library PLM XML file, and open the file. Note: • If you assign a Materials attribute from a loaded palette, a copy of the assigned attribute is placed onto the Current Document palette. You can make changes to, or rename, the copied attribute on the Current Document palette without affecting the original attribute on the loaded palette.

To	Do this
	Use the Load Palette option on the Palette Preferences dialog to automatically load a palette when a file is opened.
Merge palettes into the active palette	- Choose Merge Palettes. - In the Merge Palettes dialog box, select what to merge. Choose from Materials Images Texture Coordinates Lights - Click Browse, navigate to the location of the appearance palette PLM XML file, and open the file. Note: Use the Merge Palette option on the Palette Preferences dialog to automatically merge the selected palette into the palette of the next opened assembly file.
Merge one or more SVM (Siemens Visual Material) files into the active palette.	- Choose Merge SVM File(s). - In the Choose SVM Location dialog box, navigate to the desired SVM file(s) and click Open. The new SVM material(s) are placed on the active palette. Note: The SVM files are located in the SVM Materials directory in the installation path, or in the user-defined location specified in the SVM Database Location field on the 3D Loader Preferences dialog.

Specify Appearance Palette preferences

Entries on the Appearance Palette must have unique names. When working with appearance libraries, you may attempt to load a palette with entries that have the same names as entries that you have already opened or created within the application. Use the Appearance Palette preferences to specify default behavior for the resolution of Appearance Palette name conflicts.

- On the **3D Appearance** toolbar, click one of the following:

- Material Palette** 

- Light Palette** 

The appropriate page of Appearance Palette appears.

Tip:

You can enable more than one Appearance Palette at a time.

- Right-click an empty area on the Appearance Palette, and choose **Preferences**.
- In the **Palette Preferences** dialog box, change any of the following options:

On this page	Use this option	To do this
Conflict Resolution	Resolve to the existing entry (using Reference files)	If attempting to load a named Appearance Palette entry, check whether there is an existing entry on the Palette or in the Reference files with the same name. If a match is found, retain the entry currently on the Palette.
	Resolve to the new entry	If attempting to load a named Appearance Palette entry, check whether there is an existing entry on the Palette. If a match is found, replace the entry on the Palette with the new entry being loaded.
	Rename the new entry to be unique	If attempting to load a named Appearance Palette entry, check whether there is an existing entry on the Palette. If a match is found, retain the entry currently on the Palette, and load the new entry by making the name unique.
	Attempt to resolve to an existing, unnamed entry with the same content	If attempting to load an unnamed Appearance Palette entry, check whether there is an existing entry on the Palette or in the Reference files with identical content. If a match is found, retain the entry currently on the Palette.
	Set as default	Save your settings.
Appearance Palettes	Add	Add a PLM XML appearance library file to the list of referenced libraries.
	Remove	Remove a PLM XML appearance library file from the list of referenced libraries.

On this page	Use this option	To do this
	Move Up / Move Down	Change the order of the list of referenced libraries.
	Active Reference File	Enable or disable a PLM XML appearance library file that is included in the list. You can specify to include any of the following content options under the Content section: • Materials • Images • Texture Coordinates • Lights
	Load Palette	Load the selected palette when an assembly file is opened.
	Merge Palette	Merge the selected palette into the palette of the next opened assembly file.
	Set as default	Save your settings.

- Click **OK**.

Specify appearance reference files

You can specify default appearance libraries.

- Choose **File**→**Preferences**→**3D Loader**.
- On the Appearance page of the **3D Loader Preferences** dialog box, do any of the following:

This option	Does this
Add	Add a .plmxml appearance library file to the list of referenced libraries.
Remove	Remove a .plmxml appearance library file from the list of referenced libraries.
Move Up / Move Down	Change the order of the list of referenced libraries.
Active Reference File	Enable or disable a PLM XML appearance library file that is included in the list. You can specify to include any of the following content options under the Content section:

This option	Does this
	• Materials • Images • Texture Coordinates • Lights
Load Palette	Load the selected palette when an assembly file is opened.
Merge Palette	Merge the selected palette into the palette of the next opened assembly file.
SVM Database Location	Identifies an alternate location for the SVM (Siemens Visual Material) database files. During installation, the SVM files are installed in the default location, which is the SVM Materials directory in the installation path. After installation, you can move the SVM files to a location of your choice and identify that location in this field. When upgrading Lifecycle Visualization to a newer version, you do not need to re-install the SVM files, unless there is a new SVM database.

Use the assembly to control appearance attributes

Appearance icons displayed in the assembly tree enable you to control the assignments of materials, texture coordinates, and lights.

In the **Assembly** view, do one of the following:

- Double-click an appearance icon.
- Right-click an appearance icon, and select any of the following options:
 - **Edit Material**
 - **Assign Materials**
 - **Edit Texture Coordinate**
 - **Assign Texture Coordinates**
 - **Edit Light**
 - **Assign Lights**

Overview of the assembly appearance icons

Icons representing appearance attribute assignments appear in columns in the **Assembly** view. Use the assembly to **control the assignments** of materials, texture coordinates, and lights.

Appearance icons in the assembly tree convey the following information about their state and type:

This icon	Represents
	A basic material.
	An advanced material.
	A Siemens Visual Material (SVM).
	An occurrence-specific material assignment.
	A material assigned to sub-part geometry (shapes).
	A material assigned to sub-part geometry (faces).
	Suppressed material assignments (gray).
	A material attribute assigned to hidden leaf structure (basic material icon shown).
	Cylindrical texture coordinates.
	Cubic texture coordinates.
	Planar texture coordinates.
	Spherical texture coordinates.
	Custom texture coordinates are texture coordinates that were created in the JT file from within another application, such as NX.
	A texture coordinate attribute assigned to hidden leaf structure (planar texture coordinate icon shown).
	Directional light.
	Point light.
	Spot light.
	Light map.

This icon	Represents
	A light that is off (directional light icon shown).
	A light that is on (directional light icon shown).
	An appearance attribute in the Appearance Editor (directional light icon shown).
	An appearance attribute selected in the Appearance Palette (directional light icon shown).
	A light assigned to hidden leaf structure (directional light icon shown).

Note:

- When working with parts with materials assigned to **sub-part geometry**, a material icon is displayed for each assignment.
- When working with parts that contain multiple **light assignments**, a light icon is displayed for each assignment.
- **Modifying appearance attributes** via leaf structure icons edits the actual materials, lights, or texture coordinates that are assigned to the hidden leaf structure. To view leaf structure, right-click in the assembly tree and select **Show Leaf Structure**.

Insert 2D images into 3D environments

You can insert 2D images into your 3D scenes. 2D images are automatically mapped to dynamically created, flat pieces of geometry, which appear behind your 3D models in the Viewing window.

1. On the **3D Appearance** toolbar, click **Image Palette** .
2. On the Image page of the Appearance Palette, right-click an empty area, and select **New**.
3. In the **New Image** dialog box, load a 2D raster image.

The image appears on the Appearance Palette.

Tip:

You can open multiple images at once. In the **New Image** dialog box, browse to your files, and press the Shift key on your keyboard while selecting multiple files.

4. On the Images page of the Appearance Palette, do either of the following:

To	Do this
Insert a single image	Right-click an image, and choose Insert 3D. In the Viewing window, the image appears behind your 3D model, mapped upon a flat piece of geometry. The geometry maintains the aspect ratio of the image, and is sized and positioned as a backdrop for the visible parts. In the Assembly view, the image appears as a new part beneath the Images node.
Insert multiple images	a. Press the Shift key on your keyboard while selecting the images that you want to insert. b. Right-click any of the selected images, and choose Insert 3D. In the Viewing window, the images appear mapped to flat pieces of geometry. The geometry is positioned so that it appears to be stacked, with each part offset to be visible.

- (Optional) Use part transformation tools to reposition an image.

Apply a color range to a model

You can apply a range of color to a model in the Viewing window, automatically assigning a different color to each part in order to distinguish between them more easily.

- On the **3D Appearance** toolbar, click **Color Range Tool** .

The color range preview appears at the top of the **Color Range Tool** dialog box. The left and right color spheres define the start and end of the color range.

- (Optional) In the **Color Range Tool** dialog box, change the color range in one of these ways:

To define the range using	Do this
The Appearance Editor	a. In the Materials section of the Color Range Tool dialog box, double-click the left or right spheres that show the current start and end color settings. -or- Click the left or right spheres to select them, and click Edit. b. On the Material page of the Appearance Edit dialog box, set new material color values.

The Appearance Palette	- On the Materials page of the Appearance Palette, select a material. - In the Materials section of the Color Range Tool dialog box, click the left or right spheres to select them. - In the Materials section of the Color Range Tool dialog box, select Material. - Click Get.
Parts in the Viewing window	- Select the part from which you want to copy a material. - In the Materials section of the Color Range Tool dialog box, click the left or right spheres to select them. - In the Materials section of the Color Range Tool dialog box, select Part. - Click Get.

3. Assign the color range in one of these ways:

To assign colors like this	Do this
To selected parts	- Select the parts in the Viewing window or in the Assembly view. - In the Assignments section, on the right side of the dialog box, choose Selected.
To visible parts	- In the Viewing window, display only the parts that you want to include in the color range assignment. - In the Assignments section, on the right side of the dialog box, choose Visible.
To all parts	In the Assignments section, on the right side of the dialog box, choose All .

4. Click **Apply**.

The color range is applied to the model.

Tip:

Each time you apply the color range to the model, the color assignments are randomized.

5. (Optional) Click **Restore** to restore the previous material assignments.

Note:

After you close the **Color Range Tool** dialog box, you can use the **Undo** command to undo the last color range assignment action.

Specify appearance preferences

You can specify appearance preferences for the handling of warning messages, how newly created lights are assigned to geometry, and how the Appearance Palette displays palette entries.

1. On the **3D Appearance** toolbar, click **Appearance Preferences** .
2. In the **Appearance Preferences** dialog box, change any of the following options:

Select	To do this
Show Delete Confirmation dialogs	Display a confirmation dialog box when you attempt to delete an Appearance Palette entry that is assigned to parts.
Show License Warning dialogs	Display a warning message when you attempt to enable functionality not available in your product configuration.
Show Default Light Warning dialogs	Display a warning message when you attempt to set a light using the World Coordinate System as a default light.
Show Display Warning dialogs	Display warning messages.
Show Cast Shadows Warning dialogs	Display messages informing you that you must enable shadow rendering in the Performance dialog box.
Show Multiple Assignments Warning dialogs	Display a warning message when you attempt to make an assignment to a part with more than one occurrence in the model.
Show Assembly Level Material Warning dialogs	Display a warning message when you attempt to edit a material that is assigned to a part or subassembly within a containing assembly which has an assembly-level material assignment. The warning message dialog box gives you option to edit the selected material or the assembly-level material. Note: Editing an assembly-level material affects the displayed materials of all parts within the assembly.

Select	To do this
Show Quick Color Warning dialogs	Display a warning message when you attempt to turn on the Appearance Editor from within the Quick Color Tool. Note: You cannot work with the Quick Color Tool and the Appearance Editor simultaneously. If you turn on the Appearance Editor from within the Quick Color Tool, the Quick Color Tool closes.
Editing a part's material only affects that part	When you edit a part's material that is also assigned to other occurrences of the part, specify for the changes to affect only that single part.
Assign new lights to all geometry (Models)	When you create a new light, automatically assign it to all of your models.
Use Bounding Box selection style	Specify whether to use bounding boxes to indicate which parts are selected when editing part appearance, or to use the same selection style you use for other operations.

3. Click **OK**.

13. Generating reports

Generating reports overview

The Reports Wizard generates reports that present data in a table-oriented format. Use the Reports Wizard to format a report, specify filters to apply on the data, and preview the output.

Access the Reports Wizard from specific features, such as cross section point sheets, callouts and symbols, and collaborative review and markup.

In the Reports Wizard, you:

- Specify content that appears in the report, such as headers, columns, footers, and filters.
- Preview the report by viewing the report data prior to generating it.
- Specify the report format by choosing the format and location for the report output.

Specify content that appears in the report

Specify settings for the following sections of the Reports Wizard:

- Report header: specify the header fields that appear at the top of the report and the values those fields contain.
- Report columns: specify the column headings that appear in the report.
- Report filters: specify the data filters to apply during the generation of the report data. Use filters to limit the scope of generated data in the report output.
- Report footer: specify the footer fields that appear at the bottom of the report and the values that those fields contain.

Note:

- The report definition page does not appear for all features that use the Reports Wizard.
- Some buttons may be disabled if you do not have permission to edit a field or if the option is read-only.

Define the report header

To	Do this
Add header fields	1. Click New. The Add Custom Field dialog box appears. 2. Enter a name for the header field. 3. Enter a value that the header field will contain. 4. Click OK.
Edit header fields	1. Click Edit. The Edit Field Value dialog box appears. 2. Edit the value that the header field contains. 3. Click OK.
Delete header fields	1. Select a header from the list. 2. Click Delete.
Reorder header fields	1. Select a header from the list. 2. Click Move Up or Move Down. The header moves within the list. Note: The order in which the fields appear in the dialog box indicates the order in which they will appear in the report data.

Define report columns

To	Do this
Select columns	1. Select one or more tables to appear in the report. 2. Select one or more checkboxes, which represent column headings in the report.

Define report filters

To	Do this										
View the filter definition	1. Select a filter from the list. 2. Select View.										
Edit the filter	1. Select a filter from the list. 2. Click Edit. The Edit Custom Filter dialog box appears. Note: You cannot edit the name of an existing filter. <table data-bbox="441 842 1464 1197"> <tr> <th>Use this option</th><th>To do this</th></tr> <tr> <td>Field</td><td>(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.</td></tr> <tr> <td>Comparison</td><td>(Optional) Select how the filter will behave.</td></tr> <tr> <td>Value</td><td>(Optional) Either enter a value or select from the drop down list to filter the selected column on.</td></tr> </table> 3. Click OK.	Use this option	To do this	Field	(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.	Comparison	(Optional) Select how the filter will behave.	Value	(Optional) Either enter a value or select from the drop down list to filter the selected column on.		
Use this option	To do this										
Field	(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.										
Comparison	(Optional) Select how the filter will behave.										
Value	(Optional) Either enter a value or select from the drop down list to filter the selected column on.										
Add a custom filter	1. Click Add. The Add Custom Filter dialog box appears. <table data-bbox="441 1444 1464 1852"> <tr> <th>Use this option</th><th>To do this</th></tr> <tr> <td>Name</td><td>Enter a name for a new filter.</td></tr> <tr> <td>Field</td><td>(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.</td></tr> <tr> <td>Comparison</td><td>(Optional) Select how the filter will behave.</td></tr> <tr> <td>Value</td><td>(Optional) Either enter a value or select from the drop down list to filter the selected column on.</td></tr> </table>	Use this option	To do this	Name	Enter a name for a new filter.	Field	(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.	Comparison	(Optional) Select how the filter will behave.	Value	(Optional) Either enter a value or select from the drop down list to filter the selected column on.
Use this option	To do this										
Name	Enter a name for a new filter.										
Field	(Optional) Select a column on which to filter. This list is populated from the available column list in the report definition.										
Comparison	(Optional) Select how the filter will behave.										
Value	(Optional) Either enter a value or select from the drop down list to filter the selected column on.										

To	Do this
	2. Click OK .
Remove a filter	1. Select a filter from the list. 2. Click Remove . 3. Click OK .

Define a report footer

- Do one of the following:

To	Do this
Add footer fields	- Click New. The Add Custom Field dialog box appears. - Enter a name for the field. - Enter a value for the footer field. - Click OK.
Edit footer fields	- Click Edit. The Edit Field Value dialog box appears. - Enter a value for the field. - Click OK.
Delete footer fields	Click Delete.
Reorder footer fields	- Select a field from the list. - Click Up or Down to move the field within the list. Note: The order in which the footer fields appear in the list dictates how they will appear in the report.

2. Click **Next**.

The **Reports Preview** page appears.

Preview reports

View the report data prior to generating the full report. This makes it easier to change report options before you create the report.

Note:

The preview reports page does not appear for all features that use the Reports Wizard.

1. From the list, select a preview option.

The data in the **Preview** dialog box changes to reflect your selection.

Note:

The display grid only contains one table at a time. For reports that contain multiple tables, select each table from the list to see a preview.

2. Click **Next**.

The **Report Output** dialog box appears.

Specifying report output and presentation

Specify the output format and location for the report output. Choose either standard or custom for the report output format.

Standard report output types have predefined stylesheets. A stylesheet for each standard output type is defined in an .xsl file in the **/Visualization/etc/ReportTemplates** folder within the installation directory.

If you have your own stylesheet (.xsl file) for reports, for example: *MyCompany.xsl*, then you choose the custom report output format.

Choose a standard report output format

1. Select **Standard report output format**.
2. From the **Select Format** list, choose one of the available formats, which have associated stylesheets.
3. Click **Browse** to find and select the location where you want to save the report.

4. (Optional) Select **Automatically open report output file** to open the file in the associated application when you click **Finish**.
5. Click **Finish**.

Choose a custom report output format

1. Select **Custom report output format**.
2. In the **Stylesheet Location** box, click **Browse** to find and select the location of your stylesheet (.xsl file).
3. Specify the location of the report output by doing the following:
 - a. Click **Browse** to find and select the location where you want to save the report.
 - b. Add the extension to the file name.

Tip:

The file name extension depends on the type of output file defined by your stylesheet, for example: .htm, .txt, or .csv.

4. (Optional) Select **Automatically open report output file** to open the file in the associated application when you click **Finish**.
5. Click **Finish**.

14. Editing parts

Part simplification overview

You can view, create, and delete part Level of Details (LODs). New LODs are created by tessellating from B-Rep data present in the file, or by simplifying an existing LOD. A modified part can be exported as a new .jt file.

Note:

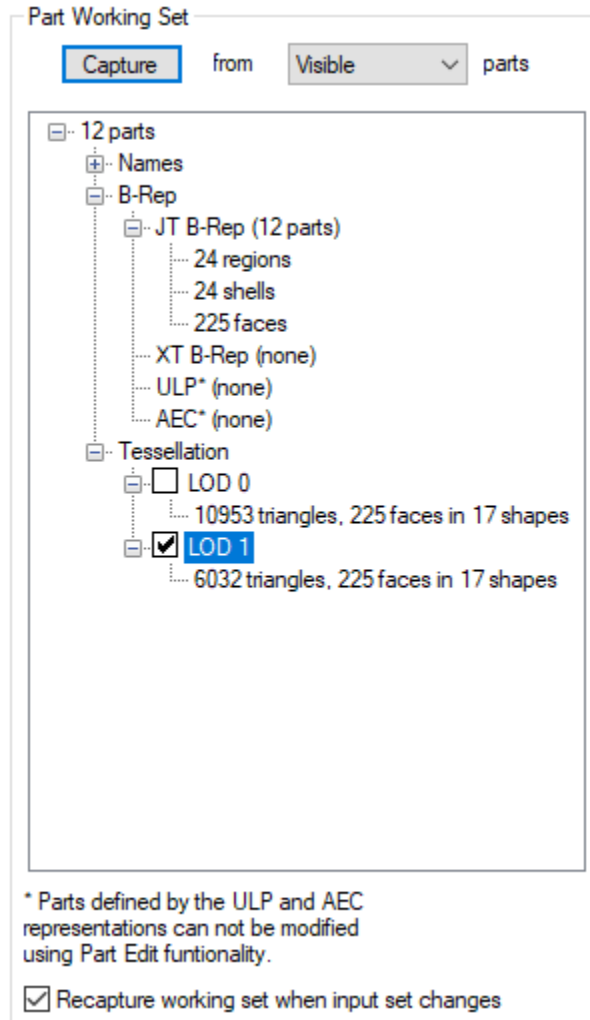
- The Part Edit functionality requires an optional module for you to export edited models. You can view model LODs and simplify parts if you do not have a license, but you will not be able to save your changes.
- Session files do not support Part Edit data. To save your work when editing parts, you must export your model as a new .jt file.

Overview of the Part Working Set

When editing or creating part LODs, you can define a working set of parts that will be affected by your actions. Specify parts for inclusion in the working set on the Control page of the Part Edit dialog box. You can have only one working set at a time, regardless of the number of open 3D documents or windows in the application.

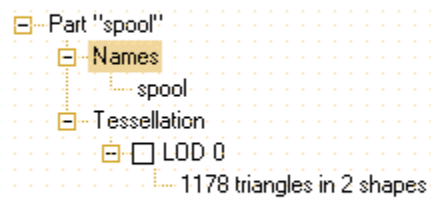
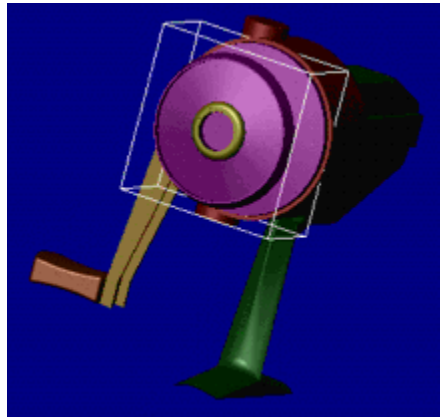
You can define a working set of parts based upon **Selected** parts in the 3D Viewing window or in the **Assembly** view, **Visible** parts in the 3D Viewing window, or **All** of the parts present in a model. The contents of the working set tree are updated according to your selection.

The top level node in the tree provides you with the part name (if the working set consists of a single part) or the number of parts in the set. The second level in the tree expands to display part names, B-Rep information, and the LODs present in the set.



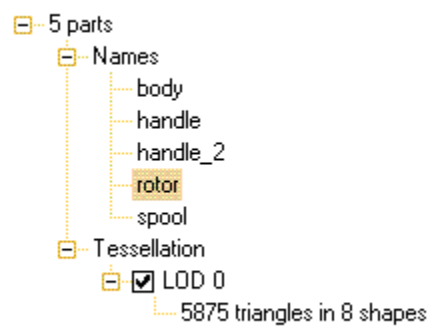
Create a working set of parts to view model LODs.

If you select **Recapture working set when input set changes**, you can dynamically create new working sets. If you choose to capture the working set from **Visible** or **Selected** parts, you can adjust part visibility or select different parts and the contents of the working set will change according to your actions.



Create a working set by selecting a part in the 3D Viewing window.

Selections in the working set tree also result in corresponding part selections in the Viewing window.



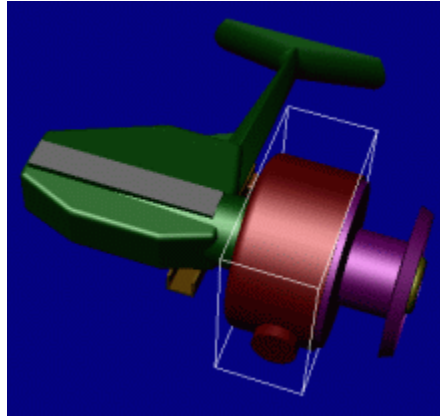


Figure 14-1. Highlight parts by selecting the contents of the working set tree.

The **Tessellation** sub-level displays all of the LODs present in the model. Selecting an LOD (by placing a checkmark in the associated box) updates the contents of the 3D Viewing window for all of the parts in the working set. Parts that are not in the working set are displayed according to the **3D Performance settings**. If no LOD is selected, the Viewing window display is also based on current 3D Performance settings.

Note:

If the working set contains parts with different numbers of LODs, then the LOD entries that are not shared are displayed in the tree as **LOD X (Y parts)**. For example, if there is one part with two LODs, and one part with three LODs, then the third LOD node reads **LOD 2 (1 part)**. If the display is locked into the third LOD, then the first part displays nothing in the Viewing window, as it does not have a third LOD.

LOD entries in the working set tree display the number of triangles, line segments, points, or primitives (cube, sphere, cone) that are contained in that LOD. The numbers are a summation of all the numbers over all of the part instances that are contained in the working set. For example, if a working set contains two instances of a part with 300 triangles in one shape, then the displayed number of triangles is 600.

Define the Part Working Set

When editing or creating part LODs, you define a working set of parts that will be affected by your actions.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. On the **3D Part Edit** toolbar, click **Control** .
3. (Optional) On the Control page of the **Part Edit** dialog box, select the **Recapture working set when input set changes** check box.

4. In the **Part Working Set** section, do any of the following actions:

To	Choose this option	Then do this
Define selected parts as the Part Working Set	Selected	the parts that you want to capture as a working set.
Define visible parts as the Part Working Set	Visible	In the 3D Viewing window, display only the parts that you want to capture as a working set.
Define all of the parts in the model as the Part Working Set	All	No action necessary.

5. Click **Capture**.

The contents of the working set tree are updated to reflect your selection.

Manage part names and LODs

You can sort parts in the working set tree by name, or by the number of triangles, lines, points, or shapes in the highest LOD. When sorting by the number of triangles, lines, points, or shapes, the number is shown in front of each part name in parentheses.

- On the **3D Part Edit** toolbar, click **Enable Part Edit** .
- On the **3D Part Edit** toolbar, click **Control** .
- In the Part Working Set tree, do one of the following:

To	Do this
Sort the part names in the working set	Right-click Names and choose one of the following options: • Sort alphabetically • Sort by LOD 0 area • Sort by LOD 0 triangles • Sort by LOD 0 avg. triangle area • Sort by LOD 0 lines • Sort by LOD 0 points

To	Do this
	• Sort by LOD 0 shapes
Delete B-Rep data from the model	Expand the B-Rep tree entry, right-click a B-Rep type and choose Delete .
Sort the LODs in Part Working Set	Right-click a LOD and choose one of the following options: • Move to top • Move up • Move down
Copy/paste or delete LOD shapes	Right-click a tree entry in the form of "X triangles in Y shapes" and choose Copy or Delete. To paste shapes into another LOD, after copying the shapes, right-click an LOD and choose Paste. Note: • To paste LOD shapes, the target LODs must consist of single parts. • You can mix the copy/paste operations on shapes and LODs. For example, you can select Copy on a shape tree entry and then select Paste After on an LOD line. • When working with multiple LODs and deleting shapes or entire LODs, it is possible to lose track of entities with which you are working. For example, suppose your Part Working set contains several parts, and there is one LOD for which some parts contain only triangles shapes, and other parts contain triangle and line shapes. If you delete the triangle shapes in this LOD, the whole LOD will be deleted for the parts that contain only triangle shapes, but it will remain for the other parts (now containing only the line shapes). This may result in a surprising update of information for this LOD, and all following LODs.
Copy/paste LODs	Right-click an LOD and choose Copy. To paste an LOD into another LOD, after copying an LOD, select the LOD that you want to paste the data into, and then click Paste Before or Paste After. Note: To paste LODs, both the source and the target LODs must consist of single parts.
Delete a LOD	Right-click an LOD and choose one of the following options:

To	Do this
	• Delete • Delete to bottom Note: When you right-click the Tessellation entry and select Delete, all LODs in the working set are removed.

Managing LOD labels

You can create labels for the LODs in your model.

Create an LOD label

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. On the **3D Part Edit** toolbar, click **Control** .
3. On the Control page of the **Part Edit** dialog box, in the Part Working Set tree, right-click an LOD and choose **Set label**.
4. In the **LOD label** dialog box, type a name.
5. Click **OK**.

Modify an LOD label

1. In the Part Working Set tree, right-click the LOD that you want to rename and choose **Set label**.
2. In the **LOD label** dialog box, type a name.
3. Click **OK**.

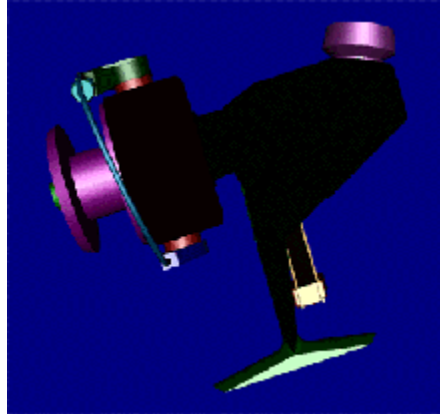
Delete an LOD label

In the Part Working Set tree, right-click the LOD that you want to delete and choose **Delete label**.

The label is deleted.

Overview of B-Rep face orientation

For parts that contain B-Rep data, you can reverse the orientation of B-Rep faces, either by individually picking the faces of editable parts (parts that contain B-Rep data), one by one, or by continuously picking faces while holding down the left mouse button and moving your cursor over the parts. When you exit Face Editing mode, the resultant high-quality LOD becomes the first LOD in the working set.



Flipped faces are displayed as black.

Tip:

To help identify faces to be reversed, disable **two-sided lighting**.

Note:

Tessellation parameters are based on the settings in the specify part edit preferences.

Reverse B-Rep face orientation

You can reverse the orientation of B-Rep faces.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. (Optional) **Define** a Part Working Set.
3. On the **3D Part Edit** toolbar, click **B-Rep** .
4. On the B-Rep page of the **Part Edit** dialog box, in the **Input** section, select one of the following options:
 - **Selected**

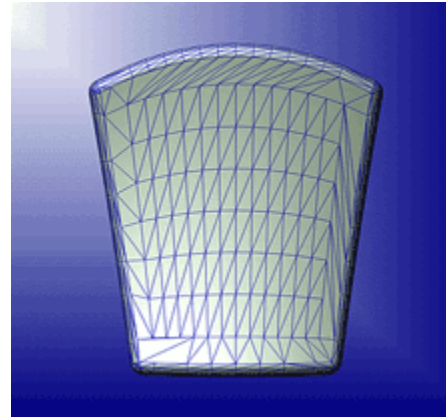
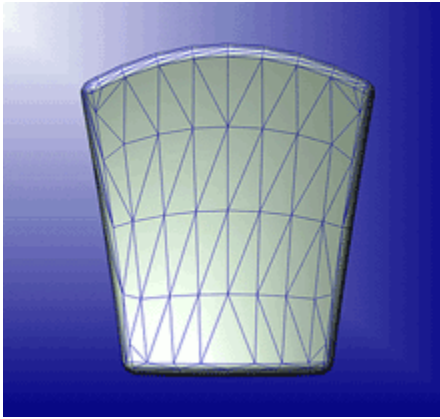
- **Visible**
 - **All**
 - **Working Set**
5. For **Reverse Face Orientation**, select one of the following options:
 - **Individual picks**
 - **Continuous picks**
 6. Click **Start**.
 7. (If you selected **Individual picks**) In the 3D Viewing window, pick the faces that you want to reverse.
 8. (If you selected **Continuous picks**) Press Ctrl and wave your mouse cursor over the model.
 9. Click **End**.

A high-quality LOD is added as LOD 0 to the model

Overview of part tessellation

If at least one part in your model contains B-Rep data, you can modify existing LODs or create new LODs by tessellating from B-Reps.

The Tessellation page of the **Part Edit** dialog box contains options for creating lines as **Curves**, as **Faces**, and as **Curves, Faces** together. Depending on your selection, the application of the tessellator will produce line shapes for B-Rep trim curves, triangle shapes for B-Rep faces, or both. When you click **Apply**, the tessellator loops over all unique parts in the input set, takes the B-Rep for each part, tessellates it with the given parameters, and places the results into a new or existing LOD.



The image on the left illustrates the number of triangles present in a part before tessellating additional triangle shapes. The image on the right is of the same part, but after tessellating.

You can preview how parameter settings will affect tessellation results. Specify the parameter settings you want to preview in the **Parameter** section. When **Precompute** is clicked, all tessellations in the specified parameter range are computed according to the **Tessellation** options (**curves**, **faces**, or **curves and faces**), and the slider is enabled. You can then use the slider to inspect tessellation results in the 3D Viewing window. The text at the bottom of the **Preview** field indicates the number of triangles or line segments for the current slider position.

Tessellate parts

For parts containing B-Rep data, you can modify existing LODs or create new LODs by tessellating from the B-Rep data.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. (Optional) **Define** a Part Working Set.
3. On the **3D Part Edit** toolbar, click **Tessellation** .
4. On the Tessellation page of the **Part Edit** dialog box, in the **Input** section, select one of the following options:
 - **Selected**
 - **Visible**
 - **All**
 - **Working Set**
5. In the **Output** section, choose to create a **New LOD** or to modify an **Existing LOD**.

6. (If creating a new LOD) Specify where in the order of existing LODs you want your change to appear.
7. (If modifying an existing LOD) Choose the LOD that you want to modify.
8. In the **Tessellation** section, specify Parameter values.
 - absolute: specify tessellation parameters in absolute units (document units, inches)
 - relative: specify tessellation parameters relative to each individual part's size (bounding box diagonal length)

For other settings, see the specify part edit preferences page.

9. From the **Tessellate** drop down box, select one of the following:
 - Curves: Create line shapes for B-Rep trim curves.
 - Faces: Create line faces as triangular shapes for B-Rep faces.
 - Curves, Faces: Create line faces as both B-Rep trim curves and triangular shapes for B-Rep faces.
10. Click the **Advanced** button to access the Minimum segment length feature, which specifies the minimum length allowed.
11. (Optional) Click **Preview**, specify the parameters you want to preview, and then click **Precompute**.

The preview slider is enabled. Use the slider to inspect tessellation results in the 3D Viewing window. The text at the bottom of the **Preview** section indicates the number of triangles or line segments for the current slider position.

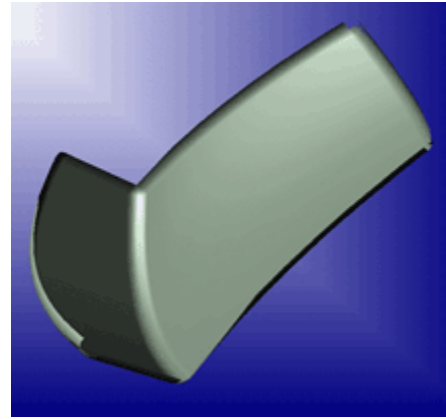
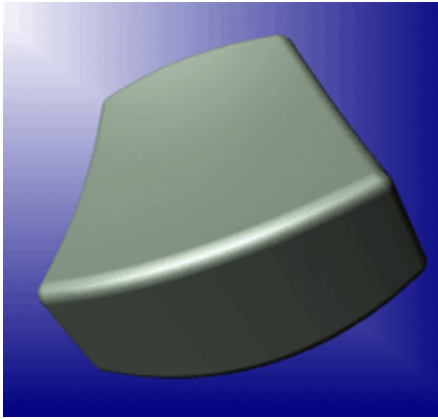
Click **Set** to populate the **Tessellation** section with your settings.

12. Click **Apply**.

The parts specified in the **Input** section are tessellated with the specified parameters, and the results are placed into a new or existing LOD.

Overview of how to simplify parts based upon visibility

You can define LODs based upon part visibility in the 3D Viewing window, retaining triangles that are visible from a specified viewpoint or set of viewpoints. For usage scenarios where only the outside shape of a complex model is required, you can also "shrink wrap" your model by collecting all of the triangles that remain after visibility simplification as a new, single model that is inserted into the Assembly.



This part was simplified by removing all geometry not visible from the viewing angle shown in the image on the left. The image on the right is of the same part, but rotated to display the part modifications.

Viewpoint options include **Current viewpoint**, **Captured viewpoints**, and **Viewpoints on a surrounding sphere**. You can select any combination of options, create a series of evenly distributed viewpoints, and then add extra viewpoints to cover difficult areas that would otherwise show as holes in the output.

Visibility is determined using one of four granularities, shapes, triangle strips, individual triangles, or B-Rep faces, with a unique color assigned to each entity of the chosen granularity. For shapes, triangle strips, and individual triangles, pixels are rendered with a unique color, and a pixel with that unique color is considered visible if it appears within any specified viewpoint.

Tip:

Because visibility is determined per pixel, maximize pixel resolution and work within a 3D Viewing window that is as large as possible.

Note:

Visibility simplification requires 32-bit color.

Simplify parts based upon visibility

Define LODs based upon part visibility in the 3D Viewing window.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. (Optional) **Define** a Part Working Set.
3. On the **3D Part Edit** toolbar, click **Visibility** .

4. On the Visibility page of the **Part Edit** dialog box, in the **Input** section, select one of the following options:
 - **Selected**
 - **Visible**
 - **All**
 - **Working Set**
5. In the **Input** section, select the LOD that you want to simplify.

Note:

- This option is available only if you choose to modify existing tessellation. This option is not available if you choose to create new tessellation from B-Rep data in your model.
- The list of available LODs to modify is limited to the number range shared by all parts of the input set.

6. In the **Output** section, choose one of the following options:

To	Do this
Create a new LOD	Select New LOD , and specify where the LOD will appear in the working set tree.
Modify an existing LOD	Select Existing LOD , and choose the LOD that you want to modify.
Collect all of the triangles that remain after visibility simplification as a new, single model that is inserted into the Assembly	Select New inserted model , and enter a name for the new model. Note: New models created using Part Edit are optimized for rendering performance. Due to this optimization, using these models to perform analysis operations will not provide the same quality of results as the original model.

7. In the **Viewpoints** section, choose one or more of the following options:

To	Do this
Capture the contents of the 3D Viewing window as a viewpoint	Select Current viewpoint .
Capture multiple viewpoints	a. Select Captured viewpoints. b. Click Capture Start. A viewpoint is captured, and the number 1 appears next to Captured viewpoints. c. Manipulate the contents of the 3D Viewing window by rotating, selecting snapshots, etc. As the 3D Viewing window updates according to your actions, the number of captured viewpoints increases. Tip: - Click Show to display the sequence of captured viewpoints. - Click Clear to remove all captured viewpoints from memory. d. Click Capture End.
Capture a series of evenly-spaced viewpoints on a sphere surrounding your model	Select Viewpoints on a surrounding sphere, and choose a resolution for the shrink wrap. Choose from the following resolutions: - Low (12 viewpoints) - Medium (42 viewpoints) - High (162 viewpoints) - Super (642 viewpoints)

8. In the **Granularity** section, specify the granularity to be used in the visibility simplification. You can modify the tessellation of an existing LOD (according to **Shapes**, **Primitives**, or **Individual triangles**), or create new tessellation from B-Rep data in the model.

Note:

- If you choose to create new tessellation from B-Rep data, the visibility simplification is preceded by a tessellation of all of the B-Rep faces that are found in the parts designated in the **Input** section. The settings for the tessellation are specified in the part edit preferences.
- If the number triangles exceeds the maximum number allowed during visibility simplification (24-bit or 16,777,216), select **Primitives** or **Shapes** instead of **Individual triangles**.
- B-Rep faces that are identified as invisible (not visible within a specified viewpoint) can be removed from their B-Rep by selecting **Remove invisible faces from the B-Rep**. Subsequent tessellations of B-Rep data altered in this way will contain only the remaining, visible faces.

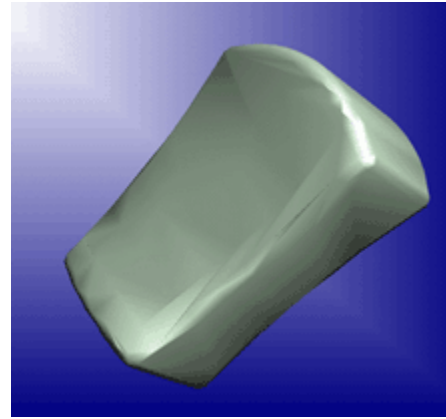
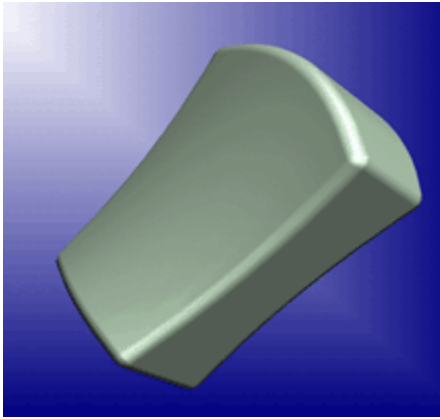
9. Click **Apply**.

Depending upon how you set up the simplification, an existing LOD is modified, a new LOD is added to the working set tree, or a new model is inserted in the Assembly.

Overview of how to reduce the part triangle count

You can reduce triangles present in LODs by defining a target percentage or entering a target triangle count, or by choosing to remove triangles with vertices that are closer than a specified distance.

Decimations that remove triangles that are closer than a specified distance collapse all vertices within the specified distance, and then remove all associated, degenerate triangles. You can specify the vertex collapse distance in absolute document units, or as a percentage of the input set's bounding box diagonal. For reference, the size of the bounding box diagonal is displayed at the bottom of the **Decimation** section.



The image on the left shows a part before decimation. The image on the right is of the same part, but after decimation. Enlarged to illustrate the effects of the decimation, the part has a "crumpled" appearance.

You can preview how parameter settings will affect decimation results. After specifying a minimum/maximum decimation range, click **Precompute** to enable the preview slider. Use the slider to inspect decimation results in the 3D Viewing window. The number of triangles that would remain if you performed the decimation using the current preview settings is displayed near the bottom of the **Preview** section.

Remove triangles

You can reduce triangles present in LODs by defining a target percentage or entering a target triangle count, or by choosing to remove triangles with vertices that are closer than a specified distance.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. (Optional) **Define** a Part Working Set.
3. On the **3D Part Edit** toolbar, click **Decimation** .
4. On the Decimation page of the **Part Edit** dialog box, in the **Input** section, select one of the following options:
 - **Selected**
 - **Visible**
 - **All**
 - **Working Set**
5. In the **Output** section, choose to create a **New LOD** or to modify an **Existing LOD**.

6. (If creating a new LOD) Specify where in the order of existing LODs you want your change to appear.
7. (If modifying an existing LOD) Choose the LOD that you want to modify.
8. In the **Decimation** section, choose one of the following options:

To	Do this
Reduce the triangle count by defining a target percentage or entering an absolute target triangle count	a. Select Target. b. Enter a target percentage or a target triangle count.
Remove triangles with vertices that are closer than a specified distance	a. Select Remove triangles with vertices closer than. b. Enter the percentage of each part's bounding box or the size of the largest bounding box within which triangles with vertices will be removed.

9. (Optional) Click **Preview**, specify the parameters you want to preview, and then click **Precompute**.

The preview slider is enabled. Use the slider to inspect decimation results in the 3D Viewing window. The text at the bottom of the **Preview** section indicates the number of triangles for the current slider position.

Click **Set** to populate the **Decimation** section with your settings.

10. Click **Apply**.

Depending upon how you set up the decimation, an existing LOD is modified or a new LOD is added to the working set tree.

Use the Visibility and B-Rep filters

Use the Visibility and B-Rep filters to constrain which parts are loaded or displayed in the 3D Viewing window.

1. Choose **Actions→Filters**.
2. In the **Filter Manager** dialog box, from the **Available Filters** list, select one of the following filters:
 - **Visibility Filter**
 - **B-Rep Filter**

3. Click **Add**.

The filter is added to the **Filter Sequence** list. This list determines the order in which filters are applied to the model assembly.

4. In the **Filter Sequence** list, select the check box for the filter.
5. From the **Filter Input Select** list, select one of the following options:

To filter these parts	Select this option
All parts in the assembly, whether or not they are loaded	All Parts
Parts that are loaded, whether or not they are displayed	Loaded Parts
Parts that are displayed	Visible Parts
Parts that are selected	Selected Parts
Parts that pass the previous filter in the Filter Sequence list.	Previous Output

6. From the **Filter Output Action** list, select one of the following options:

To have this effect on the parts	Select this option
No changes to the part state as a result of the filter	None
Display only the parts that pass the filter; hide all parts that do not pass the filter	Show Only
Display the parts that pass the filter, in addition to parts that are currently displayed	Show Additive
Hide the parts that pass the filter	Hide
Select the parts that pass the filter; deselect all parts that do not pass the filter	Select Only
Select the parts that pass the filter, in addition to parts that are currently selected	Select Additive
Deselect the parts that pass the filter	Deselect

7. Click **Properties**.
8. In the **Properties** dialog box, set the appropriate properties for the filter.
9. Select or clear the check box to the left of the filter name, if necessary.

Select the check box to apply the filter when you click **Apply**; clear the check box to disregard the filter when you click **Apply**.

10. In the Filter Manager, click **Apply** to apply the filters without closing the Filter Manager.

The filters in the **Filter Sequence** list are applied in the order that they are listed.

Specify Part Edit Preferences

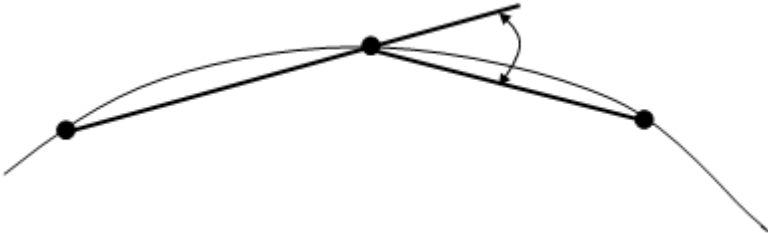
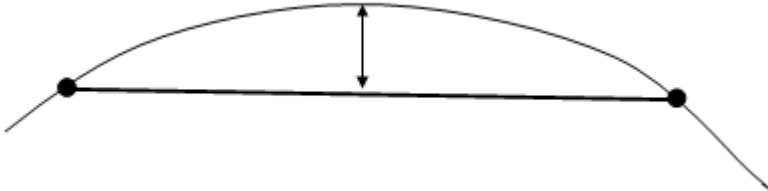
You can customize the default tessellation settings, and your changes will be saved between instances of the application. Parameters can be specified as absolute or relative values.

1. On the **3D Part Edit** toolbar, click **Enable Part Edit** .
2. On the **3D Part Edit** toolbar, click **Preferences** .
3. In the **Part Edit Preferences** dialog box, in the **Parameter values** area of the **Tessellation** section, choose **Absolute** or **Relative**.
 - **Absolute**: specify tessellation parameters in absolute units, such as document units or inches.
 - **Relative**: specify tessellation parameters relative to each part's size (bounding box diagonal length).

The tessellation parameter entries change to reflect your choice.

4. Enter a value for each preference you want to customize. You can specify values for the following settings.

Option	Description
Feature removal	Prior to tessellating, all features of the given size (feature bounding box diagonal) are temporarily removed from the B-Rep. This is useful for reducing the resulting tessellation's triangle count. For example, small drill holes in the geometry of a mechanical part often account for a good fraction of the overall triangle count
Maximum Segment Angle	Two consecutive line segments in the tessellation of a curve form an angle, as shown below. This parameter specifies the maximum angle allowed (smaller angles force finer tessellations). Tip: This refers to line segments created as approximations to a given curve. A line segment is created if it violates any one of the three criteria.

Option	Description
	
Maximum chordal deviation	A line segment in the tessellation of a curve approximates the curve with a certain chordal deviation, as shown below. This parameter specifies the maximum deviation allowed (smaller values force finer tessellations). Tip: This refers to line segments created as approximations to a given curve. A line segment is created if it violates any one of the three criteria.
	
Maximum segment length	A line segment in the tessellation of a curve has a certain length, as shown below. This parameter specifies the maximum length allowed (smaller values force finer tessellations). Tip: This refers to line segments created as approximations to a given curve. A line segment is created if it violates any one of the three criteria.
	
Minimum segment length	A line segment in the tessellation of a curve has a certain length. This parameter specifies the minimum length allowed.

5. In the **Visibility** section, type the frame delay value for showing viewpoints in **sec frame delay when showing viewpoints**.

If the model contains more than one viewpoint, use **sec frame delay when showing viewpoints** to specify how long the application looks at data from each viewpoint. You can use this delay value when you want to create a visibility simplification by calculating what geometry to keep and what to remove from particular viewpoints around the model.

6. Click **OK**.

Your settings are saved.

15. Undoing actions

Overview of undoing actions

You can use the **Undo** command to undo many common actions.

You can undo:

- 3D selection actions.
- 3D visibility actions.
- Part transformations.
- Appearance changes.
- 3D markup actions.
- Deletions of 3D snapshots, individual 3D measurements, groups, alternate hierarchies, coordinate systems, motion paths, and cross sections.

Undo the previous action or multiple actions. You can also redo actions that you have undone.

Note:

- **Undo** does not undo every type of action.

For example, you cannot undo navigation actions. (You can use **Previous View** to return to the previous view.)

If the previous action cannot be undone, the action is skipped and the action before the previous action is undone.

For example, if you turn off a part and then rotate your view; when you use **Undo**, the part reappears, but your view does not change.

- Some actions that appear to be single actions may require you to use **Undo** more than once.

For example, if you select two parts (by holding down Ctrl) and choose **Selected Off**, both parts turn off.

If you use **Undo**, the parts turn on *and* they are reselected. Using **Undo** again deselects the second part. Using **Undo** a third time deselects the first part.

- There is a limit to how many actions you can undo.

The default undo *stack* is 100 actions.

You can set the **TCVIS_MAX_UNDOREDO_STACK_SIZE** system variable to change the number of available undo actions. A bigger stack size may cause performance and memory issues for large models.

- Some actions clear the undo stack.

For example, if you insert a model and then later delete it, the undo stack is cleared.

Undo the previous action

- Press Ctrl+Z.

Note:

To reverse an undo, press Ctrl+Y.