



TEAMCENTER

Product Data Visualization on Active Workspace

Teamcenter 2412

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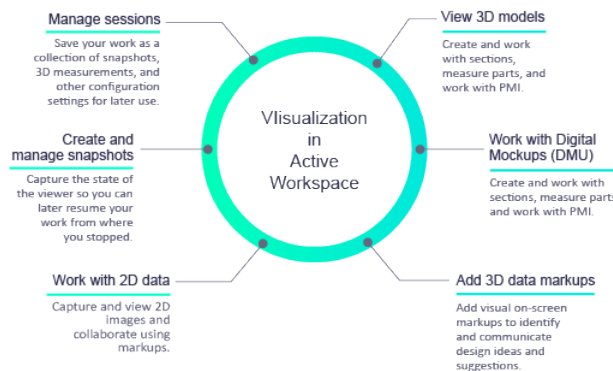
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1. About Product Data Visualization on Active Workspace



Teamcenter lifecycle visualization enables you to work with managed visualization data. You can simplify the lifecycle process of your product design by quickly and easily accessing data, analyzing it, and collaborating on your findings. Active Workspace gives you the ability to view the 3D models (CAD models translated from their native CAD-specific formats to the .jt format). The .jt files are much more efficient to work with enabling you to work with even the most complex assemblies quickly and easily.

Based on the service level available in your organization, you can perform various analytical and review tasks, including the following:

- View and explore model data.
- Get geometric data about entities of parts in an assembly.
- Measure 3D models.
- Create cross sections.
- Capture snapshots of the 3D data associated with a product.
- Add graphic on-screen markups to identify and communicate design ideas and suggestions.
- Create sessions to save the state of the viewer so you can later resume your work from where you stopped.



Where do I go from here?

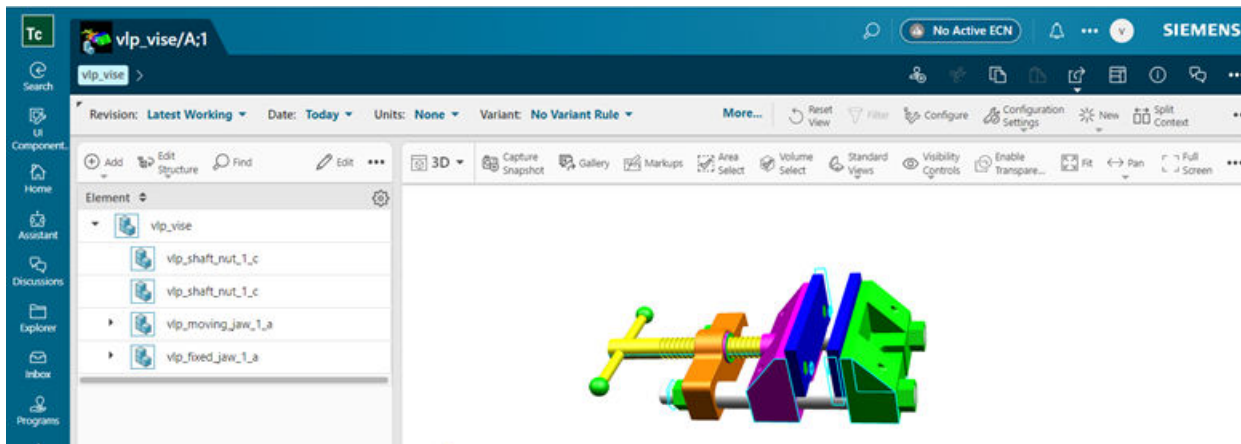
 Administrator	See <i>Visualization Deployment and Administration</i>
 Business User	

What are the different types of analysis that I can perform on models?	You can take multiple measurements of 3D models or get geometric data about entities (points, vertices, edges, faces or surfaces, and arc centers) of parts in an assembly.
How do I collaborate with other users using my 3D data?	You can use snapshots to capture sharable standalone views of an assembly or workset , or capture a series of snapshots of a single sharable assembly within a session .
How do I display the product and manufacturing information (PMI) saved within a model?	You can view PMI for a model in the 3D viewer.
Can I view a cross section of the model?	Yes, you can pass a plane through your model and display a cross section of the model where it is cut by the plane.
How do I communicate design ideas and suggestions with colleagues?	When working with assemblies in the 3D viewer, you can add graphic on-screen markups to the model in the 3D viewer.

2. The 3D viewer

In Active Workspace, the 3D viewer helps you explore JT data associated with parts and assemblies. You can use it to manipulate views, measure parts and features, view part and assembly level product and manufacturing information (PMI), create sessions to save the state of the viewer, create 3D sections through your data, and view and manage Clearance results in Active Workspace.

You can also use the viewer to capture and markup 2D images of your 3D data to support collaborative workflows such as issue capture and change requests.



3. Features of Visualization service levels

The different pages and panels used for visualizing 3D data are available to you based on your Visualization service level license: Base, Standard, Professional, or Mockup.

Features included in the Base service level

Fit, show, hide, and select	Snapshot capture
Standard views	PMI
Pan, zoom, and rotate modes	Open in Visualization
Select, cross-select, multiselect	Hosting in viewer and AWConnect
Measure	Transparent selection toggle
Geometric Information	Part coloring
Section	Search results only
2D View and Markup	Standard Views
Area Select	Snapshot and Image Galley
Split Views	Explode
Image view	Show only results
Render acceleration	Activate logging to collect debug information (only available to administrators)
Viewer Options	

Additional features included in the Standard service level

- Image capture
- 2D and 3D Markups
- True Shade material preview

Additional features included in the Professional service level

- Part positioning

Additional features included in the Mockup service level

- Clearance results
- Proximity filter
- Volume filter

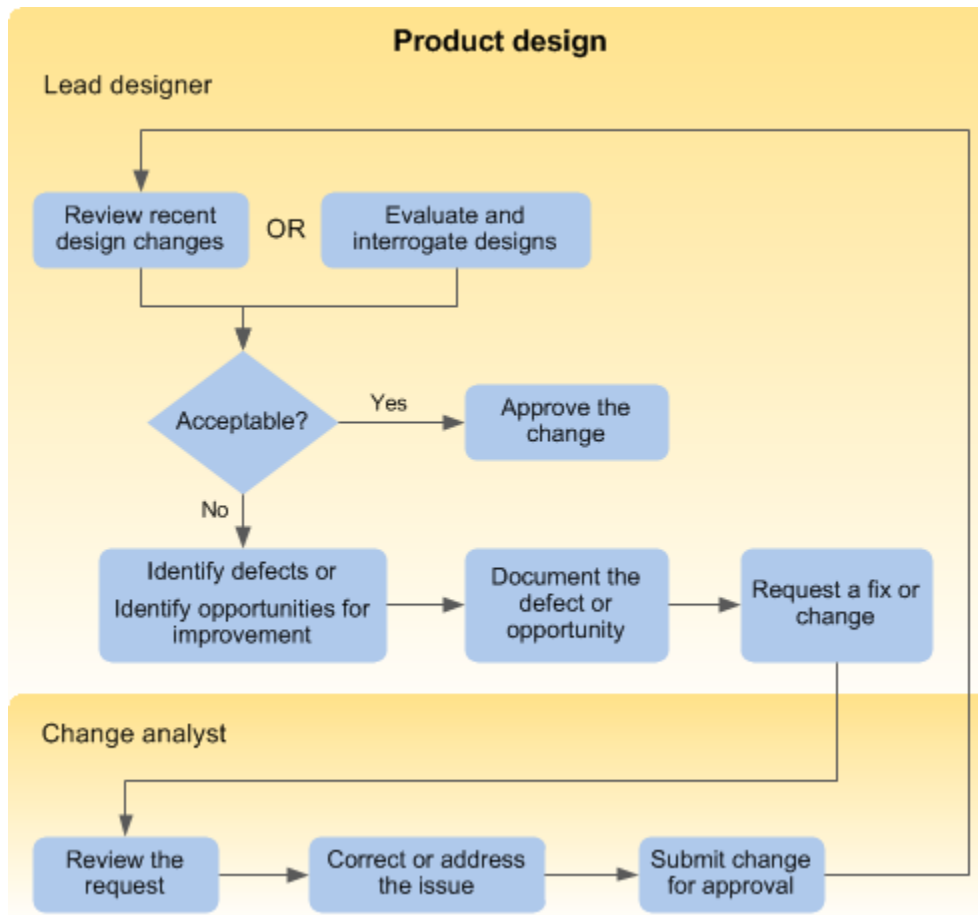
Note:

Massive Model Visualization (MMV) is an optional, add-on feature available only with Professional and Mockup service level licenses.

4. 3D viewer business processes

Design: 3D viewer business process

In the *product design* business process, people use the 3D viewer to examine the quality of the design or respond to requests for change.

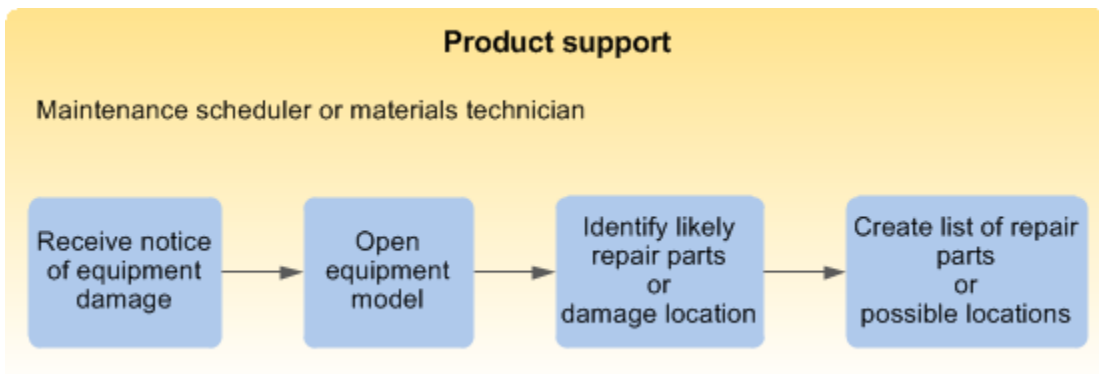


Business process step	Description	Performed by role
Review recent design changes or Evaluate and interrogate designs	Without need for the CADD software, examine a 3D model to visualize an individual part or an assembly.	Lead designer
Identify defects or opportunities for improvement	Review the design to identify defects or opportunities for improvement.	Lead designer
Document the defect or opportunity	Capture images and your mark-ups of the model.	Lead designer
Request a fix or change	Create a request for a fix or change.	Lead designer

Business process step	Description	Performed by role
Review the request	Examine the request, including attached images with markup.	Change analyst
Correct or address the issue	Make changes to the design in the CADD software.	Change analyst
Submit change for approval	Document the new design and send it for approval.	Change analyst
Approve the change	Review the change and approve it, or send it for additional re-work.	Lead designer

Service desk: 3D viewer business process

In the *product support: service desk* business process, the maintenance scheduler or materials technician uses the 3D viewer to find parts and part numbers within a model.

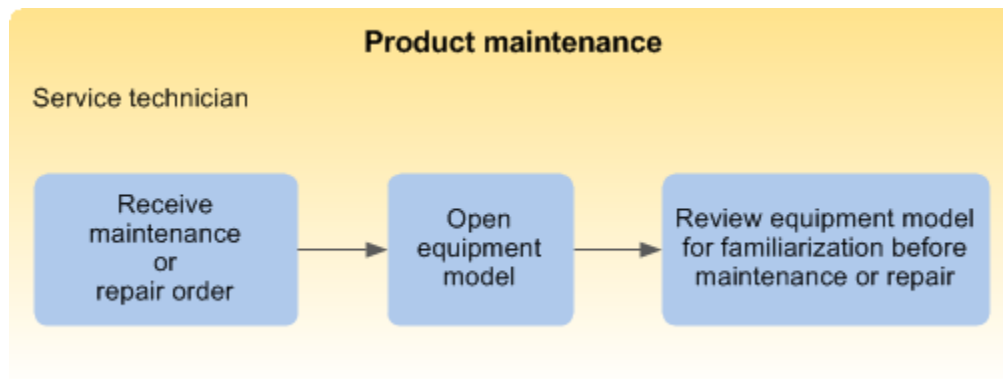


Business process step	Description
Receive notice of equipment damage	View the notice containing information about the damage, such as the location where the damage was sustained, or a found part. Scenario 1: Someone ran a forklift into an aircraft door. What parts are involved? Scenario 2: An aircraft collided with a bird. What should the assembly look like before the collision? Scenario 3: We found a part loose on the ground. Where might it have come from?
Open the equipment model	Enter the aircraft tail number or other equipment identifier to load the correct model in the viewer.
Identify likely repair parts or damage location	Identify the parts that might need repair. Either of the following may apply: <ul style="list-style-type: none"> You know the part location in the assembly, but not its number.

Business process step	Description
	<ul style="list-style-type: none"> You have the part and its number but do not know its exact location in the assembly. For instance, the part has 10 occurrences, and you want to know the locations to check for missing parts.
Create a list of repair parts or possible locations	<p>Do one of the following, as appropriate:</p> <ul style="list-style-type: none"> Create a list of parts needed to have on hand in order to ensure that the equipment can be properly repaired. Be careful to not over-specify or under-specify needed parts. Create a list of locations to be checked for damage.

Repair: 3D viewer business process


In the *product maintenance: repair* business process, the service technician uses the 3D viewer to become familiar with an assembly before performing maintenance or a repair.



Business process step	Description
Receive maintenance or repair order	You are directed to perform maintenance or a repair.
Open equipment model	You open a model of the equipment in its ideal state.
Review equipment model for familiarization	You manipulate the model to refresh your familiarity with the appearance of the components and their assembled appearance.

5. Open a 3D model




In Active Workspace, you can view your model (part or assembly) in the **3D** viewer.

You can choose to view specific parts of the model by selecting those parts from the tree. You can use the **All On command** in the **Visibility Controls**  list to view the entire model. Your administrator defines whether or not the entire model is loaded into the viewer when the model is opened.

Note:

The 3D viewer enables the visualization of 3D objects that are linked to an item revision using an *IMAN_Rendering* relationship. If you attempt to associate other object types, for example, a PDF using *IMAN_Rendering* relationship, then such objects will not load and will display an error in the Active Workspace 3D viewer.

Additionally, when multiple *IMAN_Rendering* relationships are created within an item revision, the 3D viewer chooses and displays one of the objects, in no specific order.

1. Search for and open a part or an assembly that has associated 3D model (JT) data.
2. (Optional) Prior to displaying your model in the viewer, you can choose whether your model should be rendered on the client or on the server. Click **More Commands**  from the work area toolbar and select **View**  > **Viewer Options** .
3. In the **Viewer Options** panel that opens, under **Render Options**, select the desired **Model Render Location**:

- For server-side rendering: **Server**

Note:

A valid graphics card must be available on the Visualization Server for **Server** to be enabled.

- For client-side rendering (default setting): **Client**

Note:

A valid graphics card must be available on the client machine and WebGL must be enabled in your browser for **Client** to be enabled.

4. Click **Close**.
5. Choose **Details** > **3D** to display the model in the viewer.

As you wait for the product structure to load, you can see a progress bar appear at the top of the 3D workspace. Click **Stop** on the right side of the progress bar to interrupt the loading of the product structure. The progress bar is displayed when you attempt to load data that requires any additional product structure.

Example:

Consider a scenario where the **AWC_visAllOn** user preference is initially set to *False* and only the minimal structure and no geometry is loaded in the viewer. Now, if you use the tree to make sub-assemblies or the entire assembly visible, the progress bar is displayed while the request for product structure is processed.


You will receive a warning if you load a session file that contains unresolved NGID references. If this happens, load the session file into Lifecycle Visualization and resolve the broken links. See *Resolving broken references* in the *Lifecycle Visualization* documentation.

You can right-click a part in the 3D viewer and choose the desired option:

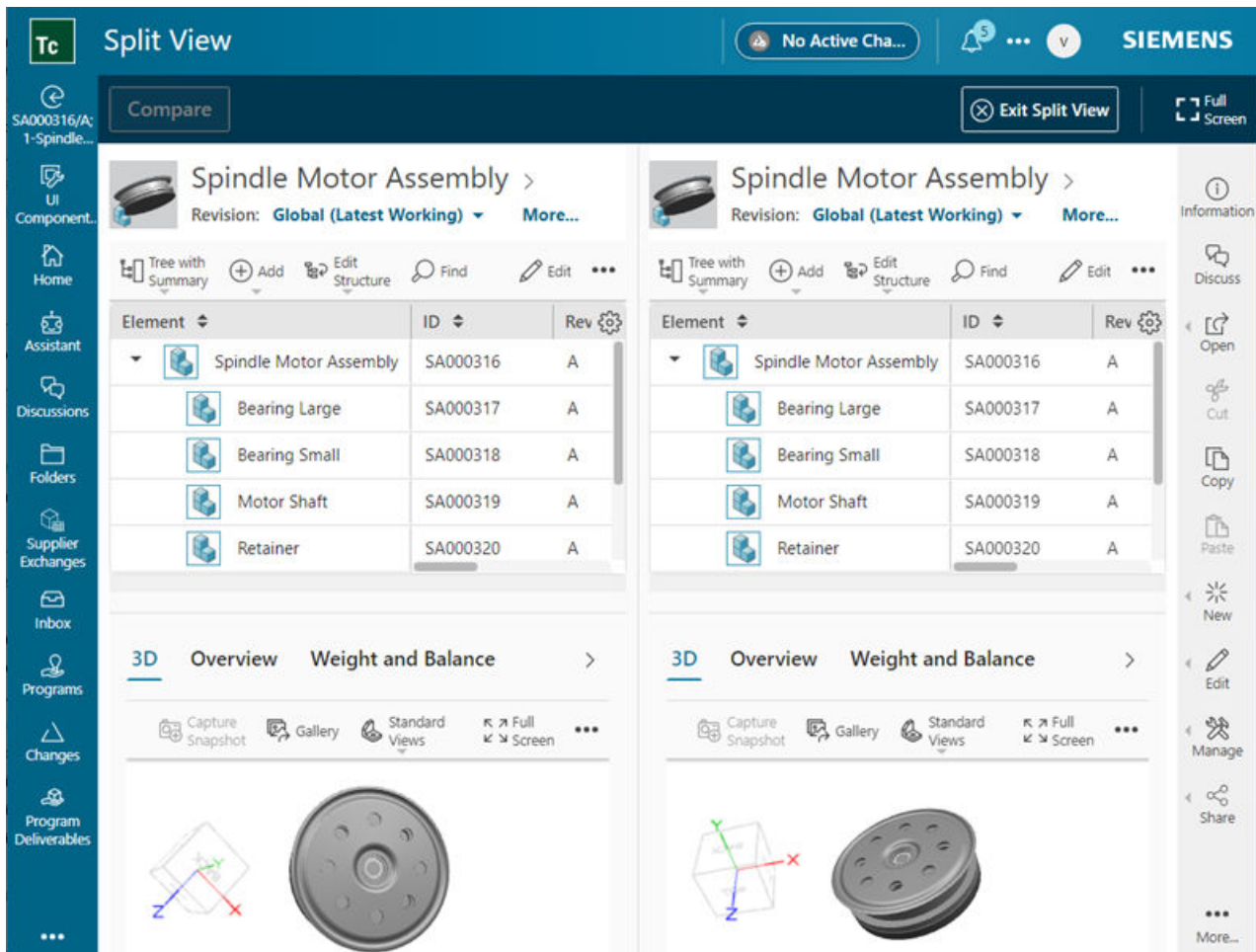
- **Show Only:** To display only the selected part and hide the remaining assembly.
- **Hide:** To hide the selected part.

You can split the context view to get into a side-by-side view of two independent structures. The two windows are independent, in terms of selection, navigation, and configuration change. You can either load the same structure in both the windows, or you can load two separate structures.

To view the same structure in both windows —

1. Search for and open the structure you want to work with.
2. Click **Split Context** .

When a structure is split into two views, using the **Split Context** command, identical structures are loaded in both the views with a 3D viewer relevant to the structure. The commands under the 3D viewer in each view work with their respective context.



Note:

When a structure is split into two views, the current state of the structure is auto-saved and loaded in the left view whereas the default state of the structure is loaded in the right view.

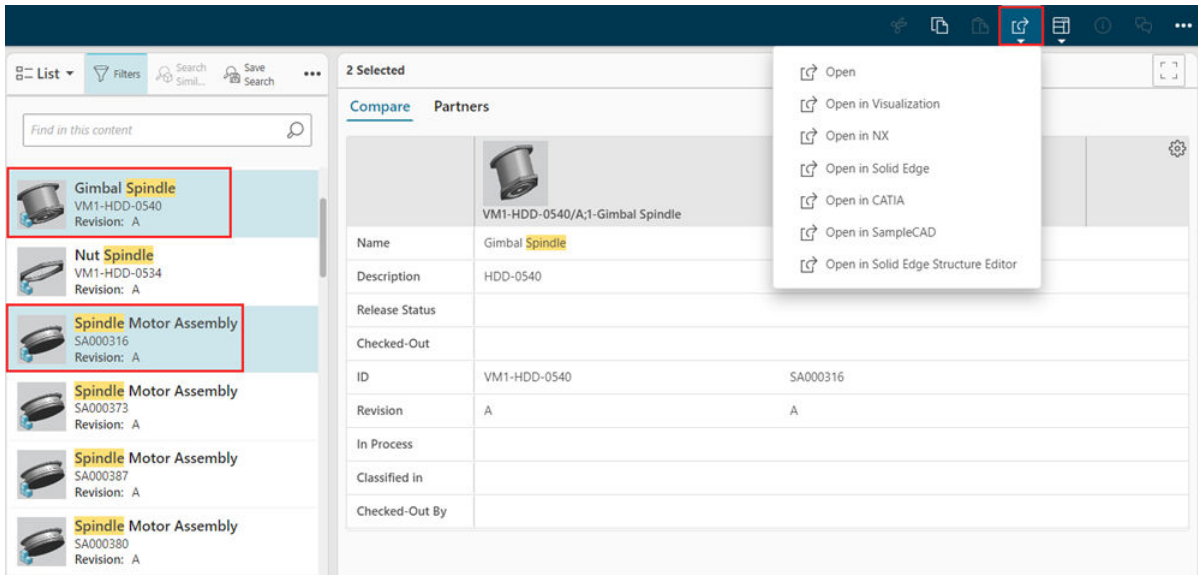
- If you get out of the split view, the auto-saved state of the structure is reloaded.
- If you refresh the browser, both views reload to display the default state of the structure.


To get out of the **Split Context** view, click **Exit Split** ⊗.

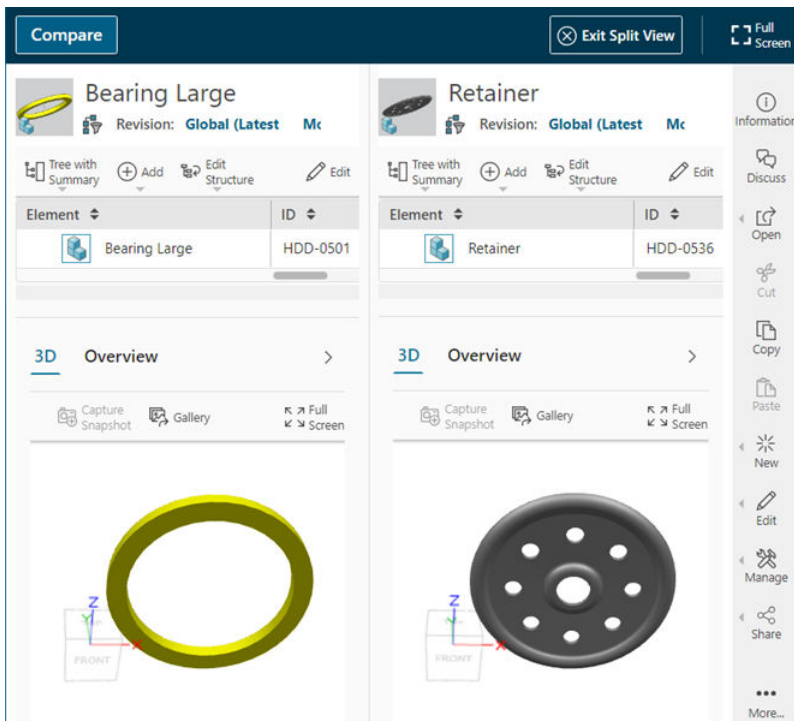
To view two separate structures in both windows —

1. From the search results, select the first structure using either **Ctrl+click** or **Right-click**. This activates the multiselect mode.
2. From the search results, select the second structure that you want to open in another window.
3. On the primary toolbar, click **Open** ↗.

4. From the drop-down list, select **Open** .



When two different structures are opened side-by-side in the split view using the **Open**  command, each structure is loaded in a separate view with a 3D viewer relevant to the structure. The commands under the 3D viewer in each view work with their respective context.



To get out of the **Split** view, click **Exit Split** .

You are re-directed to the part that was selected last.

6. 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:

This feature is not supported with MMV data.

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

7. Connecting with Teamcenter Lifecycle Visualization

Selection and visibility synchronization

When hosting Active Workspace in Teamcenter Lifecycle Visualization or when connecting through Active Workspace Application Connect, data selection and visibility is synchronized between the Active Workspace tree and the Lifecycle Visualization Project Workspace. For example, if you select one or more parts from either tree, those same parts are selected in the Viewing window (Lifecycle Visualization) or 3D viewing area (Active Workspace). Likewise, if you turn off part visibility in either tree, those same parts are no longer visible in the Viewing window (Lifecycle Visualization) or 3D viewing area (Active Workspace). Synchronization is supported at all levels, including subsets, partitions, and elements. Synchronization is also supported during Pack and Unpack.

For synchronization to work, an assembly with the same root node must be visible in both the viewer and the Active Workspace tree. To force Active Workspace to switch to the desired assembly, in the standalone viewer, right-click the root node and select **Show in Active Workspace**.

Note:

Because Active Workspace discontinued support for Internet Explorer in Active Workspace 6.0, Teamcenter lifecycle visualization 14.1 and earlier can only host Active Workspace 5.2 and earlier. Teamcenter lifecycle visualization 14.2 and later can host any supported version of Active Workspace.

Open documents in standalone Lifecycle Visualization




Use this procedure to send data to Lifecycle Visualization. If your system is configured to use Active Workspace Application Connect or hosting, sending data to the standalone viewer will let you connect your work between Active Workspace and the standalone viewer. Use preference settings to configure your system for data connection.

The parts of an assembly are sent to the standalone viewer based on their state of selection and visibility:

- If parts are selected, the selected parts are sent, even the ones that are not visible.
- If no parts are selected but some parts are visible, the visible parts are sent.
- If no parts are selected and no parts are visible, the top node of the structure is sent, causing the entire structure to be sent.

- If the user is in split view, and if parts or assemblies selected last by the user are in the left view, then only those parts or assemblies are sent. Similarly, if parts or assemblies selected last by the user are in the right view, then only those parts or assemblies are sent.

To open an object in Lifecycle Visualization:

1. Select an item revision, part, or structure and click **Open** .
2. From the Primary toolbar, click **Open**  > **Open in Visualization** .




The application generates a VVI file that contains the information necessary for the interoperability between the data in Active Workspace and the data in the standalone visualization viewer.

3. As needed, give the browser permission to open the VVI file.

Caution:

You can select a *part* from a folder or from the search results, open it in the standalone viewer, and save it within a session. However, the session cannot be interoperated between the standalone viewer and the 3D viewer in Active Workspace.

To be able to open a session for a part in the 3D viewer in Active Workspace:

1. Search for the part and select it from the search results.
2. Click **Open**  > **Content**.
3. From the primary toolbar, click **Open**  > **Open in Visualization** .

If you are connecting through Active Workspace Application Connect, the VVI file opens the data in a new tab in your default browser. If you are connecting through hosting or simply opening your data in the standalone viewer, the VVI file opens the data in the standalone viewer.

Open multiple documents in Active Workspace Application Connect

Using Active Workspace Application Connect, you can open multiple documents in the following ways:

- Open multiple documents within one Active Workspace Application Connect browser tab (one browser tab connected to multiple documents in the standalone visualization viewer).
- Open multiple documents each in their own Active Workspace Application Connect browser tab (multiple browser tabs connected to multiple documents in the standalone visualization viewer).

Note:

Your preference settings determine whether you use Active Workspace Application Connect or hosting when you select the **Open in Visualization** option. Hosting does not support opening multiple documents.

To open multiple documents —

1. **Open a document** using **Open in Visualization**.

The document is opened in Active Workspace in a new browser tab and is opened in the visualization viewer.

2. Do one of the following.

To	Do this
Open multiple documents within one browser tab.	<ol style="list-style-type: none"> a. In the new browser tab, open a new document using Open in Visualization. b. In the standalone visualization viewer, on the 3D Load Options dialog select Open document and then click OK. <p>Now there are two documents in the visualization viewer which are connected to the documents open in one browser tab.</p> c. (Optional) To sync the visualization viewer with the correct document in the browser, do one of the following: <ul style="list-style-type: none"> • In the browser tab, move to the desired document tree. • In the visualization viewer, right-click the root of the tree and select Show in Active Workspace.
Open multiple documents in separate browser tabs.	<ol style="list-style-type: none"> a. Navigate to the original browser tab where Active Workspace is running (the non-Active Workspace Application Connect tab). b. Open a new document using Open in Visualization. <p>The document is opened in a new browser tab and in the standalone visualization viewer.</p> <p>Now there are two documents in the visualization viewer which are connected to documents in two separate browser tabs.</p> c. (Optional) Continue opening new documents into new tabs.

To


Do this

Remember each time to open the document from within the original browser tab.

8. Marking up 3D data

Add markups to 3D data






When working with assemblies in the 3D viewer, you can add graphic on-screen markups to identify and communicate design ideas and suggestions. You can either create freehand markups or add markups using the predefined shapes.

1. **Load the assembly** in the 3D viewer.
2. From the work area toolbar, click **Markups** .

The **Markups** embedded toolbar is displayed in the viewer area.


Note:

You can create multiple markups of any markup type.

3. To mark up using freehand drawings:
 - a. On the **Markups** toolbar, select **Create Freehand** .
 - b. In the viewer area, use the cursor to draw freehand lines to highlight a custom area of the model and then provide your feedback.
4. To mark up using predefined shapes:
 - a. On the **Markups** toolbar, click a predefined shape of your choice: **Rectangle** , **Ellipse** , or **Arrow** .
 - b. Place the cursor at the location where you want to add the markup in the 3D viewer and draw the selected predefined shape.
 - c. Select the markup and move it as needed.
 - d. Resize the markup, if required, by selecting it and then dragging the square handles at the corners and midpoints.
 - e. Select a markup and right-click to choose the required options to either edit or delete the selection in the viewer. For **Arrow** , you can modify the **Line** style. For other predefined shapes, you can modify the shape by changing its **Edge** or **Fill** styles.


Note:


Your modifications are applied only to the first markup that you have selected. You cannot modify multiple markups.

5. To add a text markup:
 - a. On the **Markups** toolbar, select **Text** .
 - b. In the viewer area, add the desired text and press Enter.
 - c. Select the text and move it as needed.
 - d. Double-click the text to change the text characters.
 - e. Select your text and right-click to choose the required options to either edit or delete the text in the viewer. You can change its **Edge**, **Fill**, or **Text** styles.

Note:

Your modifications are applied only to the text that you selected.

6. (Optional) To anchor your markup to specific parts:
 - a. Click **Anchor**  and then select a markup type, if one isn't selected.
 - b. Place the cursor over geometry in the 3D viewer and draw the desired markup.
 - c. Only for a text markup, drag the newly created markup to view a leader line from the markup to the part. If needed, you can move the anchored markup to a specific location.

Once in the **Select**  mode, if you move the part, it moves with the markup and a leader line that connects the markup to the selected part.


Note:

You can anchor all types of markups. However, only your text markups display leader lines.

- d. For anchored text, select your text and right-click to choose the required options to either edit or delete the text in the viewer. You can change its **Text** or **Leader Line** styles.


Note:

Your modifications are applied only to the text that you selected.



- (Optional) Click **Preferences**  to change your markup preferences by changing the **Line**, **Edge**, **Fill**, **Text**, or **Leader Line** styles.

Tip:

You can also change properties for a particular markup by right-clicking it and selecting **Edit** from the context menu.


- To exit the markup mode, click **Markups**  to deselect. The markups added to the view remain visible until you **remove all markups**. When you are not in the markup mode, you can move the markups and edit them using the context menu.

Tip:

After you have added the markups, you can capture them in context with the 3D data within snapshots using the **Capture Snapshot**  command. All snapshots are saved in the **Gallery** . You can then save all the snapshots within a session, which can be shared with other users to collaborate.


Delete or hide 3D markups


Perform the following steps to remove markups added to an already **opened assembly** in the Active Workspace viewer:


- From the work area toolbar, click **Markups** .




The **Markups** toolbar is displayed.

- Delete a single or all markups as desired:

- To delete a single markup, with or without the **Markups** toolbar active, select the markup, and either right-click it and select **Delete** from the context menu, or press Delete on your keyboard.
- To clear all markups from the 3D view:
 - On the **Markups** toolbar, click **Delete All** .
 - In the confirmation message box that appears, click **Delete All**.

- To hide all markups from the 3D view without deleting them, on the **Markups** toolbar, click **Hide** .

You can redisplay the hidden markup by clicking **Hide**  again.

Note that you can also remove markups by selecting **Visibility Controls**  > **Clean 3D View**  on the work area toolbar. Alternatively, right-click in the viewer, and select **Clean 3D View**  from the context menu.

Caution:

Clean 3D View also removes all the **user modifications** such as measurements, geometric information, and sections, resets part positions and colors, and hides PMI.

9. 3D viewer modes and options

What are mouse navigation modes?

On systems with mouse pointing devices, you can navigate the 3D view of a model in either of following navigation modes.


Note:

- Mouse navigation modes and options are not available with Base Visualization.
- The **Navigation** section is available only if the active rendering mode is server-side rendering (SSR). It is currently not supported for client-side rendering (CSR).

Navigation mode	Description
Examine	In this mode, navigation behaves as if you are holding the model at a point in the center of the screen. This is the default navigation mode. <ul style="list-style-type: none">• Push (default): Zoom in by pushing the mouse or scrolling the mouse wheel away from you.• Pull: Zoom in by pulling the mouse or scrolling the mouse wheel towards you.
Walk	In this mode, navigation behaves as if you are moving relative to the model. The Walk mode helps to navigate through a large model more easily.

Set mouse navigation modes and options




On systems with mouse pointing devices, you can navigate the 3D view of a model in **different modes**.

1. **Load the product assembly** in the 3D viewer in Active Workspace.
2. From the work area toolbar, click **Viewer Options** .
3. Select one of the following options in the **Viewer Options** panel > **Navigation** > **View Control** Type:
 - **Examine**
 - **Walk**

Note:

- Mouse navigation modes and options are not available with Base Visualization.
- **Examine** and **Walk** are available only if the active rendering mode is server-side rendering (SSR). They are currently not supported for client-side rendering (CSR).

The following navigation options are available on the work area toolbar for both, **Examine** and **Walk** modes:

-  **Rotate**
-  **Pan**
-  **Zoom**

Note:

When **Viewer Options** panel > **Navigation** > **Mouse Gestures** is set to **Use NX mapping**, all view commands on the toolbar (**Pan**, **Zoom**, **Rotate**) are hidden.

Navigate geometry in the Active Workspace style of mouse navigation using work area toolbar

You can navigate the view of a 3D model by clicking and dragging the mouse cursor within the 3D viewing area. When you click mouse buttons and drag the cursor to navigate, the cursor appearance changes to indicate the active navigation gesture.

In environments where you are using both Active Workspace and NX, you may wish to change your preferred navigation style to match that of NX. To do this:

Choose **Viewer Options** > **Navigation** > **Mouse Gestures** > **Use NX Mapping**.

The Active Workspace style of mouse navigation is the default navigation style and supports two navigation modes: **Examine and Walk**.

Note:

The **Walk** navigation mode is only available when the **render location** is set to **Server** and the navigation style is Active Workspace.

Pan the camera


1. On the work area toolbar, select **Pan**  as the navigation option.

- Click the left mouse button with the cursor in the 3D viewer and drag to shift the model laterally and vertically in the 3D view.


Note:

The model moves in the same direction that you drag in the **Examine** mode and in the direction opposite to the direction that you drag in the **Walk** mode.

Zoom the model

- On the work area toolbar, select **Zoom**  as the navigation option.
- Click the left mouse button with the cursor in the 3D viewer and drag up to zoom in and drag down to zoom out to move the model either farther away or closer in the 3D view.




Rotate the view

- On the work area toolbar, click **Rotate**  as the navigation option.
- Click the left mouse button with the cursor in the center of the 3D viewer and drag in the direction in which you wish to rotate the model.


This rotates the model around the center of the viewer. If you press and hold the left mouse button while over the model, that point on the model is selected as the point of rotation. After you select the point of rotation, moving the mouse rotates the model around the selected point. In this mode, a sphere indicates the rotation point. When you release the mouse, the point is removed, and navigation returns to rotate around the center of the viewer.

Based on the initial position of the cursor in the 3D viewer, you can either rotate the model freely or constrain the rotation. To rotate freely, place the cursor in the central portion of the 3D viewer before clicking. To constrain the rotation, place the cursor near the edges of the 3D view before clicking and dragging parallel to the near edge.

The following constrained rotations are available only if **Rotate** is the active option:

- To rotate around a horizontal axis: Place the cursor near the left or right edge of the viewer () and click the left mouse button and drag vertically.
- To rotate around a vertical axis: Place the cursor near the bottom edge of the viewer () and click the left mouse button and drag horizontally.
- To rotate around an axis that is perpendicular to the center of the view: Place the cursor near the top edge of the viewer () and click the left mouse button and drag horizontally.

Additional geometry focused viewing features

To achieve this	Do this
Fit the selected parts in the work area	<ol style="list-style-type: none"> Select the desired fit option in the Viewer Options panel (accessed either through Details or 3D) view > Navigation > Fit Type: <ul style="list-style-type: none"> Spherical Fit: Fits the visible geometry in such a way that no matter how it is rotated, all parts are contained within the viewing window. Tight Fit: Fits the visible geometry as closely as possible based on the current viewing angle. This can result in geometry extending beyond the viewing window edges when viewing from other angles including rotating, using the nav cube, or applying standard views. On the work area toolbar, click Fit . <div style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p>Note:</p> <p>The viewer fits the selected parts in the 3D viewer work area even if part visibility is turned off.</p> <p>If no parts are selected, all parts for which visibility is turned on, are fit into the view.</p> </div>

Navigate geometry in the Active Workspace style of mouse navigation using mouse-keyboard combinations

You can navigate the view of a 3D model by clicking and dragging the mouse cursor within the 3D viewing area. When you click mouse buttons and drag the cursor to navigate, the cursor appearance changes to indicate the active navigation gesture.

In environments where you are using both Active Workspace and NX, you may wish to change your preferred navigation style to match that of NX. To do this:

Choose **Viewer Options** > **Navigation** > **Mouse Gestures** > **Use NX Mapping**.

The Active Workspace style of mouse navigation is the default navigation style and supports two navigation modes: **Examine and Walk**.

Note:

The **Walk** navigation mode is only available when the **render location** is set to **Server** and the navigation style is Active Workspace.

You can use the following mouse button and keyboard key combinations to pan, zoom, and rotate your 3D model regardless of the work area toolbar setting.

Navigation gesture	Use these mouse buttons and keyboard modifiers
Pan	Middle mouse button and drag Additional combination: Ctrl + Left mouse button and drag
Zoom	Middle mouse roller: You can use either of these additional combinations: <ul style="list-style-type: none"> • Shift + Left mouse button and drag up or down • Left mouse button + Middle mouse button and drag up or down
Rotate	Ctrl + Shift + left mouse button and drag

Additional geometry focused viewing features

To achieve this	Do this
Center a part in the 3D viewer	Point to a location on the part and click the middle mouse button or wheel.
Rotate around a point on a model <i>without</i> moving it to the center of the view	<ol style="list-style-type: none"> 1. Place the cursor over the point on the model you want to rotate around. 2. Press and hold the left mouse button without moving the cursor until a small sphere appears beneath your cursor. 3. Once the sphere indicating the rotation point appears, drag the cursor to rotate around that point.

Navigate geometry using the NX style of mouse navigation

In environments where you are using both Active Workspace and NX, you may wish to change your preferred navigation style to match that of NX. To do this:

Choose **Viewer Options > Navigation > Mouse Gestures > Use NX Mapping**.

In the NX mode, mouse navigation works as follows.

Note:

All view commands on the work area toolbar (**Pan, Zoom, Rotate**) are hidden in the NX mode.

To do this	Use these in NX mode
Zoom view directly	Either of these options: <ul style="list-style-type: none"> • Scroll • Ctrl + middle mouse button
Rotate a model directly	Either of these options: <ul style="list-style-type: none"> • Middle mouse button • Alt + middle mouse button <p>Note that in the default rotation navigation, the model rotates about the center of the visible parts bounding box and a sphere indicates the rotation point. When the assembly is centered in the viewer, it might appear as though the rotation is around the center point of the viewer. If the assembly is moved from the center of the screen, you can see that the model rotates around the visible parts bounding box.</p> <p>If you press and hold the middle button or the Alt key in combination with the left mouse button while over the model, that point on the model is selected as the point of rotation. After you select the point of rotation, moving the mouse rotates the model around the selected point. In this mode, a sphere indicates the rotation point. When you release the mouse, the point is removed, and navigation returns to rotate around the center of the visible parts bounding box.</p>
Pan directly	Shift + middle mouse button

Additional features and key combinations




To do this	Use these in NX mode
Select an area	Either of these options: <ul style="list-style-type: none"> • Left mouse button • Shift or Alt or Ctrl + left mouse button
Open the context menu	Either of these options: <ul style="list-style-type: none"> • Right mouse button • Shift or Alt or Ctrl + right mouse button

Navigate geometry using gestures

On your touch screen device, you can use multitouch gestures to navigate the view of a model.

Note:

Gestures have the same effect regardless of whether the **Navigation** mode is set to **Examine** or **Walk**. Additionally, all two-finger gestures are recognized together. For example, performing a spin gesture while varying the distance between fingers, simultaneously rotates and zooms the view.

To perform this action	Do this
Fit the entire model in the viewer	Double tap the background.
Pan the view	Tap the screen with two fingers, hold, and drag the fingers simultaneously in the same direction.
Rotate the view	Tap and hold with two fingers, and then drag the fingers in a spin. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note: You cannot rotate a 2D image.</p> </div>
Zoom the view	Tap the screen with two fingers, hold, and spread the fingers apart to zoom in; pinch the fingers together to zoom out. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Tip: On some touch devices, the browser itself may react to the pinch gesture by zooming the entire web page. In most scenarios, you can avoid this problem by waiting for a second before placing the second finger to perform the pinch gesture.</p> </div>
Rotate, pan, or zoom the view with a single finger	<ol style="list-style-type: none"> From the work area toolbar, ensure that the option you want is selected in the drop-down list. Navigation options for single finger dragging include: <ul style="list-style-type: none"> • Rotate  • Pan  • Zoom  With a single finger, tap and hold, and then drag.

How do standard views align with axes in the 3D viewer

Active Workspace Visualization supports three standard view alignments. These three alignments control how the standard view names, such as **TOP** or **BOTTOM** align with the X, Y, and Z axes in the 3D viewer.

The following **orientation** (alignment) options are available:

Automotive

Fixed option matches the respective industry.

Orientation	Perspective or alignment
TOP	Aligned to the Z axis, facing in the -Z direction
BOTTOM	Aligned to the Z axis, facing in the +Z direction
LEFT	Aligned to the Y axis, facing in the +Y direction
RIGHT	Aligned to the Y axis, facing in the -Y direction
FRONT	Aligned to the X axis, facing in the +X direction
BACK	Aligned to the X axis, facing in the -X direction
+Isometric	A 3/4 perspective, facing downward (+Isometric)
-Isometric	A 3/4 perspective, facing upward (-Isometric)

Aerospace

Fixed option matches the respective industry.

Orientation	Perspective or alignment
TOP	Aligned to the Y axis, facing in the -Y direction
BOTTOM	Aligned to the Y axis, facing in the +Y direction
LEFT	Aligned to the X axis, facing in the -X direction
RIGHT	Aligned to the X axis, facing in the +X direction
FRONT	Aligned to the Z axis, facing in the -Z direction
BACK	Aligned to the Z axis, facing in the +Z direction
+Isometric	A 3/4 perspective, facing downward (+Isometric)
-Isometric	A 3/4 perspective, facing upward (-Isometric)

Custom


Configurable option using Teamcenter preferences.

Orientation	Perspective or alignment
TOP	Aligned to the Z axis, facing in the $-Z$ direction
BOTTOM	Aligned to the Z axis, facing in the $+Z$ direction
LEFT	Aligned to the X axis, facing in the $+X$ direction
RIGHT	Aligned to the X axis, facing in the $-X$ direction
FRONT	Aligned to the Y axis, facing in the $+Y$ direction
BACK	Aligned to the Y axis, facing in the $-Y$ direction
+Isometric	A 3/4 perspective, facing downward (+Isometric)
-Isometric	A 3/4 perspective, facing upward (-Isometric)

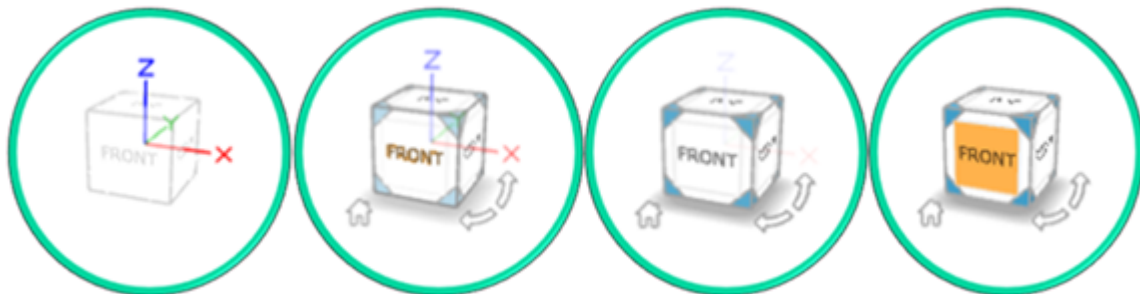
Orient the view to a standard angle

Active Workspace Visualization supports three standard view alignments to map standard view names with the X, Y, and Z axes in the 3D viewer.

To set the current view to match a standard view based on the current standard view alignment in the 3D viewer:

1. **Open the product or assembly** in the 3D viewer.
2. You can orient the view of a 3D model to a standard angle using either of these methods:
 - Click **Standard Views**  and select a view orientation, for example, **Top**.

- Use the combined navigation (Nav) cube  and trihedron  in the 3D viewer itself to select a view orientation.




The Nav cube is available only in the client-side rendering mode.

- Click the side handles to select the orientation along the axis that you want.
- Click the corners and edges of the Nav cube to get isometric views.

Note:

If you rotate the model in the 3D viewer, the Nav cube rotates to match the orientation of the model.

- Choose **Viewer Options**  > **Orientation** > desired **Standard View Alignment** from the **list**.


Note:

The **Viewer Options** command is not available with Base Visualization.

If you have split views and choose a particular standard view alignment, the same alignment is retained in both the views.

The trihedron and the Nav cube are displayed as a single entity in the 3D viewer by default. Additionally, the trihedron and Nav cube state (displayed or hidden) is saved across sessions.

To display or hide the Nav cube and trihedron in the 3D viewer:

1. Click **Viewer Options** .
2. In the **Viewer Options** panel, under **Orientation** > **Orientation Display**:
 - If you are in the client-side rendering mode: Select (or clear the selection for) the **Show Navigation Cube** check box.
 - If you are in the server-side rendering mode: Select (or clear the selection for) the **Trihedron** check box.

View CAE results in the 3D viewer

You can view JT files that contain computer-aided engineering (CAE) results in the 3D viewer, provided the following conditions are met:

- To view CAE results from a JT file, in either Teamcenter or Teamcenter Lifecycle Visualization, deactivate **Apply True Shading Materials** on the JT file. This is because applying true shading to materials overrides the CAE results node coloring.
- CAE is not supported when using client-side rendering (CSR). To view CAE data, you must set your rendering method to server-side rendering (SSR), if your server supports this capability.

To learn more about generating CAE results, see Simulation Process Management.

Change the viewer rendering method

When viewing 3D data, you can choose whether your data is rendered on the client or on the server. Client-side rendering (CSR) is typically used for smaller data sizes and may be more responsive. Server-side rendering (SSR) is more suitable for larger data sizes due to its higher data size limit. Therefore, you must use SSR to view Massive Model Visualization (MMV) data.


Note:

The default rendering method is initially defined by your administrator.

Caution:

If the current viewer rendering method is set to **Client** and you try to launch the 3D viewer for an assembly with MMV data, you get an error. In this case, update the viewer rendering method to **Server**, log out from the session, log in again, and then launch the 3D viewer for the assembly with MMV data.

To toggle between the viewer rendering methods:

1. From the work area toolbar, click **Viewer Options** .
2. In the **Viewer Options** panel that opens, under **Render Options**, set **Model Render Location** to either option:
 - For server-side rendering: **Server**

Note:

A valid graphics card must be available on the Visualization Server for **Server** to be enabled.

- For client-side rendering: **Client**

Note:

A valid graphics card must be available on the client machine and WebGL must be enabled in your browser for **Client** to be enabled.



3. On the message that displays, click **Reload** to reload your data.

Accelerate client-side rendering using occlusion culling

Occlusion culling is a method to speed up 3D rendering by not loading or rendering parts that aren't currently seen or are hidden from view. This saves memory and render time, thus enabling working with larger models when using client-side rendering.

You can accelerate client-side rendering in the 3D viewer by selecting a *render acceleration* mode that determines which parts to load or hide in the 3D viewer.

Procedure

1. **Open an assembly** in the viewer.
2. On the work area toolbar, click **Viewer Options** . If you don't find it on the work area toolbar, click **More Commands** **...** and select **Viewer Options**  from the dropdown.
3. In the **Viewer Options** panel that opens, under **Render Options**, select the render acceleration mode from the **Render Acceleration** list.

Rendering speed	Description
Standard	Loads all parts while hiding small parts when navigating.
Fast (default option)	Loads the parts that are seen and all previously seen parts in the view.
Faster	Loads only those parts that are seen in the view.
Fastest	Loads the parts that are seen in the view, while hiding the small and transparent parts when navigating.

Note:

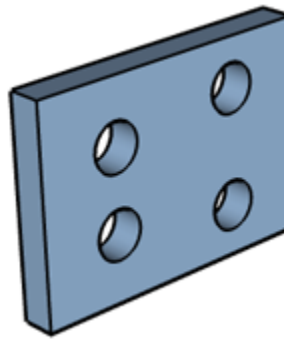
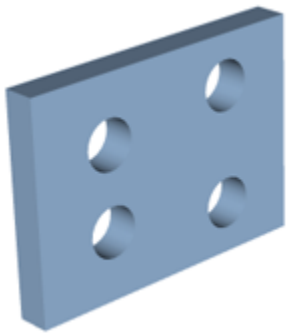
Render Acceleration is only available if you are using client-side rendering.

You can switch between **Fast**, **Faster**, **Fastest** render modes without reloading the model in the viewer. However, the model is reloaded in the viewer if you switch to either the **Standard** mode from any of the other render modes, or back to any of the other render modes from the **Standard** mode.

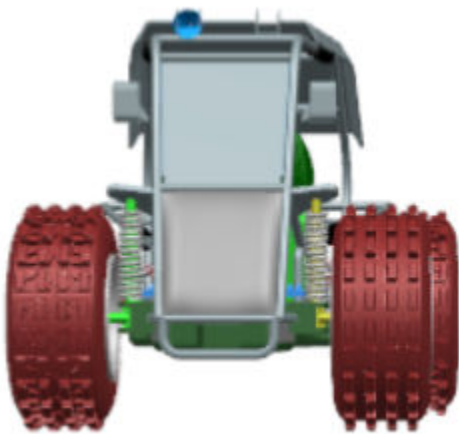
Show edges

You can choose to show edges for either of these models.

A part




An assembly



Note:

The **Shaded with Edges** option is not available with the Base level license of visualization.

1. From the work area toolbar, click **Viewer Options** .
2. In the **Viewer Options** panel that opens, under **Render Options**, select the **Shaded with Edges** check box.

Note:

The **Shaded with Edges** functionality is not available for Massive Model Visualization (MMV) assemblies.

Edges are shown on the model in the viewer.


If the model or assembly contains incorrect or broken topomesh data and if the active rendering mode is client-side rendering (CSR):

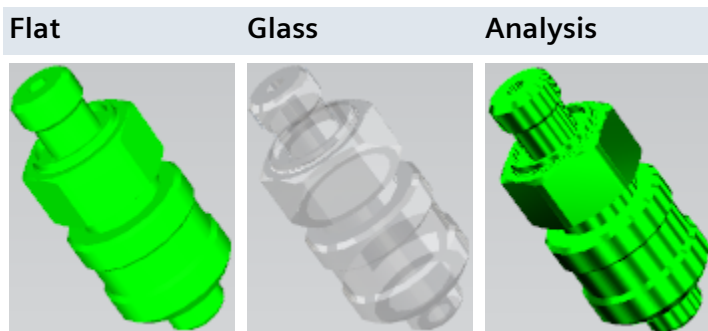
- Edges are not displayed on the model.
- An error message specifying why edges are not shown on the model is displayed.

3. To stop showing edges, clear the **Shaded with Edges** check box.


Apply material

By default, models in the viewer are displayed in shaded mode with colors or materials as saved in the model file (*Flat* material).


You can choose to apply a different material to the model (in the assembly or part viewer) to simulate the appearance of colored plastics, glass, rubber, paint, or various metals, such as chrome, copper, or steel. You can also simulate striations that indicate curvature by applying the **Analysis**  material.

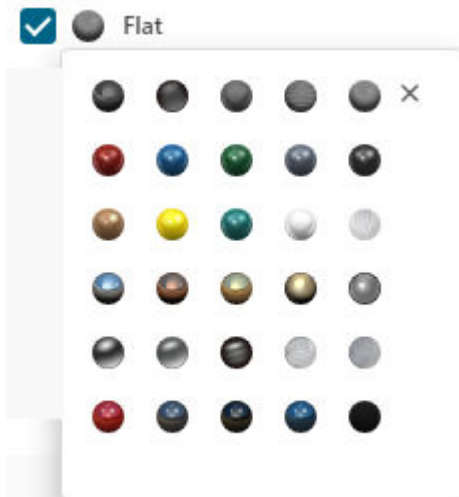


To apply a different material:

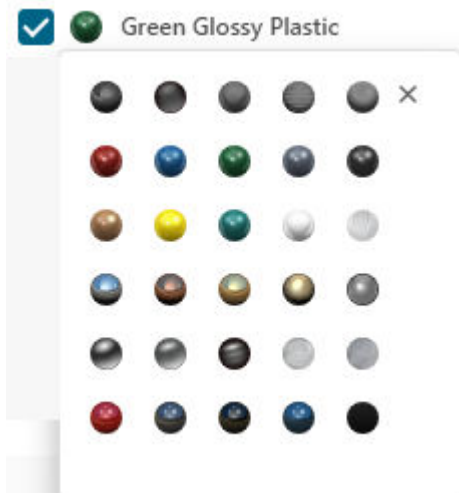
1. **Open a model** in the 3D viewer.
2. On the work area toolbar, click **Viewer Options** .
3. Under **True Shading**, click the current material icon, and then choose a material from the palette.

Note:

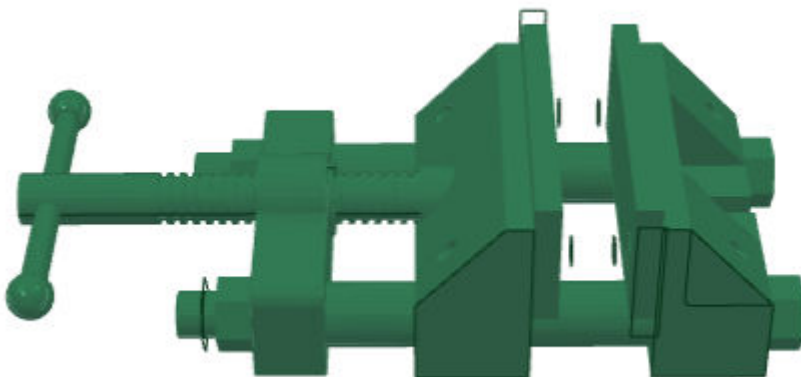
- By default, the check box for the material icon is selected. If it is cleared, you must select the check box for the material icon.
- The **Flat**  material option is available only if the active rendering mode is server-side rendering (SSR). Applying materials is not currently supported for client-side rendering (CSR) mode.



In the **Viewer Options** panel, the selected material's icon and name is updated.



In the work area, the material you selected is applied to the model.



- To turn off application of the material, clear the box next to the material icon.

Note the following:

- Your choice of material is saved across sessions.
- To see CAE results and textures, turn off the application of material.
- Applying materials is not available with Base Visualization.

Choose a color scheme in the 3D viewer

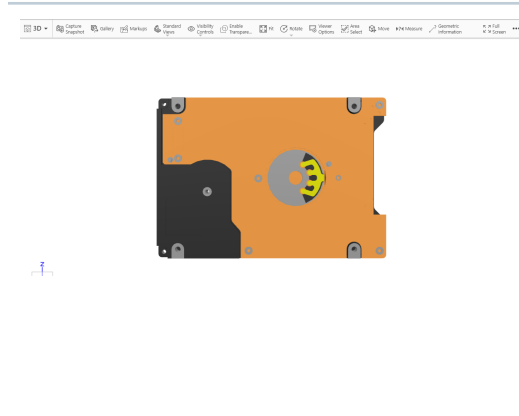
In Active Workspace, you can choose between two schemes, *White* and *Gray Ramp*, to change the background color and the part selection color in the 3D viewing window.

Color schemes

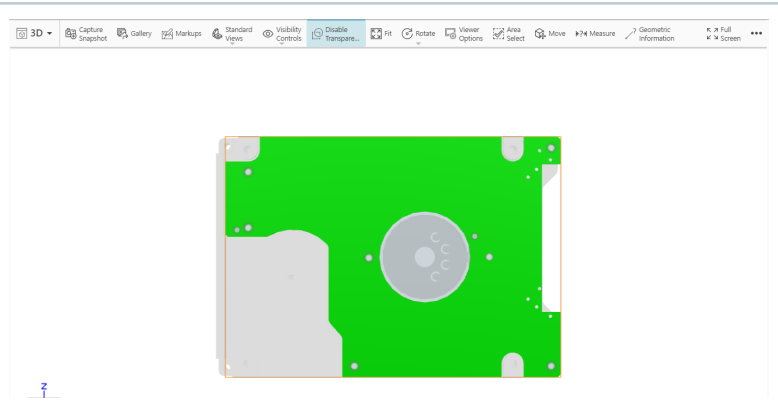
- White**

This is set as the default scheme. The background of the viewing window is set to white, and the part selection color is set to orange.

When transparency is disabled



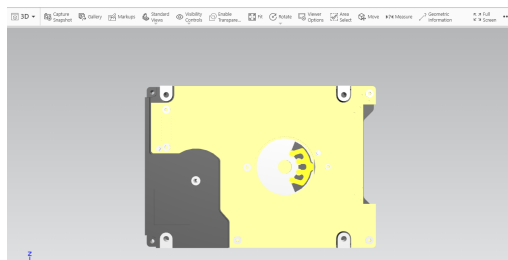
When transparency is enabled



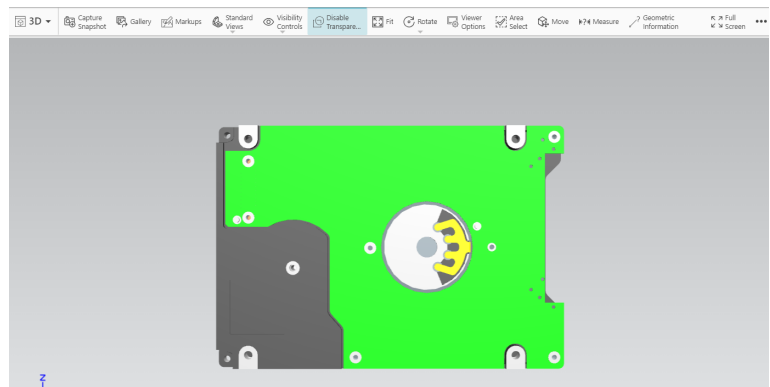
- Gray Ramp**

In this scheme, the background color is set to gray, and the part selection color is set to yellow.


When transparency is disabled

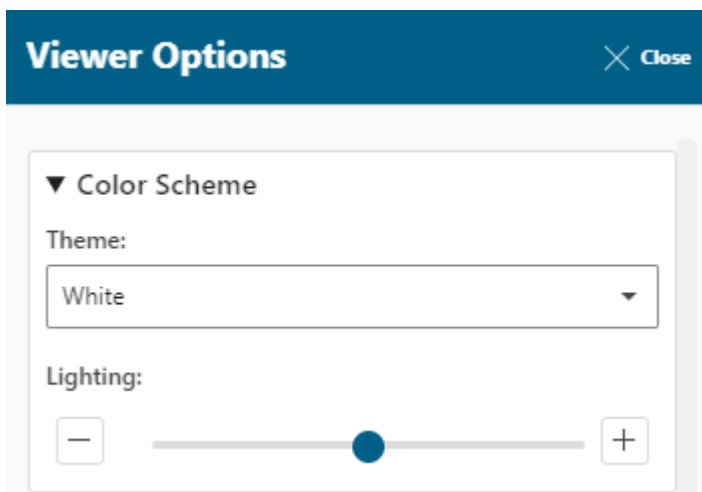


When transparency is enabled



Change a color scheme in Active Workspace

1. **Open the model** in the 3D viewer.
2. On the work area toolbar, click **Viewer Options** .
3. Under **Color Scheme**, choose the **Theme** from the list.
4. (Client-side rendering, only) Under **Lighting**, do one of the following to adjust the lighting intensity for the parts.
 - Drag the slider left or right.
 - Click the **—** or **+** buttons.





Note:

The background color and part selection color gets saved within a working session.

If you are viewing session data, the scheme is displayed from the session file. In case of **cross sections**, the selection color of the section is the same as the part selection color of the currently-active scheme. This color scheme can be the out of the box scheme or a user-defined scheme in the initialization file. Additionally, the color scheme is stored as a Teamcenter preference and is retained between sessions.

Process part information by highlighting parts

The **Highlight Parts on Mouse Hover** feature allows you to select parts more accurately and identify them more quickly by visually highlighting the part without actually selecting it. Additionally, you can view the name and ID of the part while hovering without selecting it first. Perform the following steps to disable or enable this feature, or to adjust its sensitivity.

1. **Open an assembly** in the 3D viewer.
2. On the work area toolbar, click **Viewer Options** . If you don't find it, click **More Commands ... > Viewer Options** .
3. In the **Viewer Options** panel that opens, under **Highlight**, select or deselect the **Highlight Parts on Mouse Hover** to enable or disable the feature to highlight parts when you hover over them and display the part name and ID.
4. Under **Delay Seconds**, specify the amount of time the cursor must be motionless over a component before it is highlighted in the viewer. Larger Delay Seconds values minimize unintended flickering that can result from slow cursor movements over parts.

Display a floor in the viewer


You can display a *floor* in the viewer, with effects such as grid, model shadow, and model reflection. Note that:

- The **Floor** display option is not available with Base Visualization.
- The **Floor** display option is available if the active rendering mode is server-side rendering (SSR). However, it is not currently supported for client-side rendering (CSR).
- By default, the floor is off. The floor state (on or off) is saved across sessions.





Caution:

Turning on the floor increases the load on the server and the network.

1. From the work area toolbar, click **Viewer Options** .
2. From the **Viewer Options** panel that opens, under **True Shading** section > **Floor**, select the **Show Floor** check box.

To display the floor in the viewer, select the **Grid**, **Shadow**, and **Reflection** check boxes.


You can adjust any of the following options:

Option	Action
Orientation 	Changes the orientation for the floor plane. Click to select a different orientation. The default floor orientation is XZ . The floor orientation is saved across sessions.
Offset 	Adjusts the floor position. Click and drag the slider, or click - or + to move the floor closer to or further from the model. The default floor position is 0 . The floor position is saved across sessions.
Grid	Displays a grid in the viewer.

Option	Action
	By default, the grid is displayed. The grid state (on or off) is saved across sessions.
Shadow	Displays a shadow of the model. By default, the shadow is not displayed. The shadow state (on or off) is saved across sessions.
Reflection	Displays a reflection of the model. By default, the reflection is not displayed. The reflection state (on or off) is saved across sessions.
Note:	
The reflection is not visible when the position of the navigation camera is beneath the floor (not on the same side of the floor as the model).	

Select parts in a rectangular region

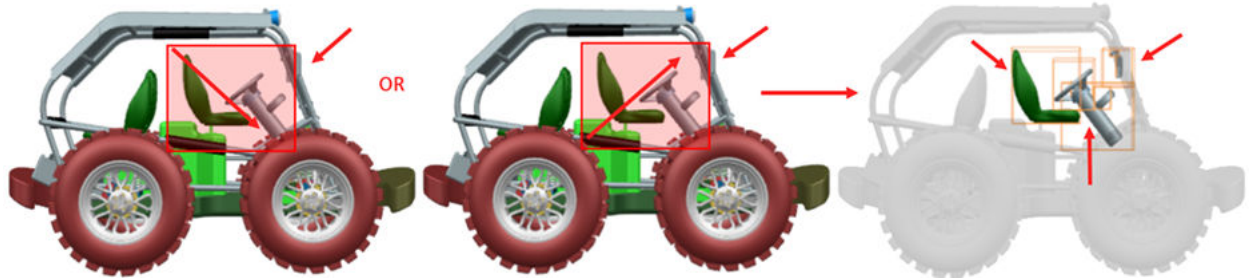
Area select speeds up the selection of multiple parts in an assembly. It enables you to indicate the parts to select by dragging a rectangular area containing or containing and crossing over those parts in the 3D viewer.

1. **Load the assembly** in the 3D viewer.
2. On the toolbar, click **Area Select** .

Note:

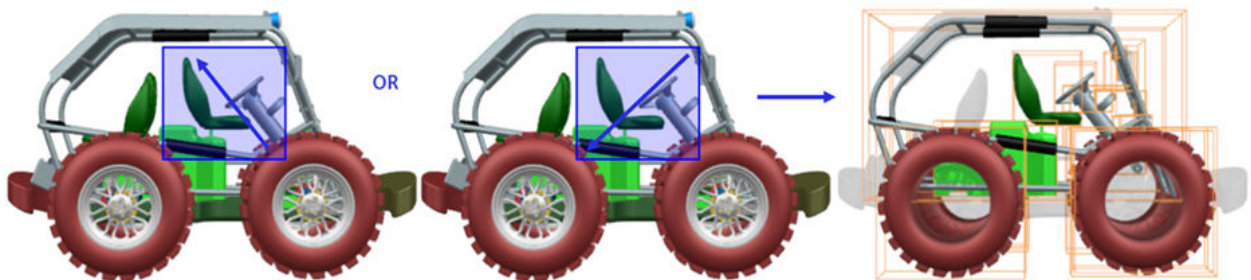
The **Area Select** functionality is not available for Massive Model Visualization (MMV) assemblies.

3. In the viewing area, click and drag to the right or left to define an area around the parts to be selected.
 - Drag to the right to select parts that are completely inside the area only. The area will have red edges when dragged to the right.

**Note:**

This action will not select sub-components of a monolithic JT in the viewer. The entire monolithic JT must be contained inside the volume to select it using the contained only mode.

- Drag to the left to select parts that are contained by or intersect with the area. The area will have blue edges when dragged to the left.

**Note:**

This action will select all components of a monolithic JT if any of the sub-components intersect the volume.

Your selections are highlighted in the 3D viewer and in the structure tree.

4. To exit from selection mode without changing the selection state, press the **Esc** key on your keyboard.

The number of parts that you can select with a single area select action is dependent on the value of the site-level preference set by your administrator. While performing an area select operation, if you exceed the maximum set limit, the following actions take place:

- A warning informing you about the current number of parts that are included in the rectangle (but not selected because they exceeded the limit) is displayed.
- The **Area Select** command stays highlighted and the viewer remains in the area select mode.
- The previous selection is retained.


- The selection which exceeded the maximum the limit is ignored.

Note:

The limit is considered only for a single *Area Select* operation. This means that the maximum limit condition is applied only to your current selection and does not apply to the total number of selected parts.

Select parts using a volume

Volume select speeds up the selection of multiple parts by dragging out a box containing, or containing and crossing over those parts in the 3D viewer.

1. **Load the assembly** in the 3D viewer.
2. On the toolbar, click **Volume Select** .

Note:

The **Volume Select** functionality is only available for client-side rendering.

3. In the viewing area, click on any geometry surface point and drag to the right or left to define a volume around the parts to be selected.

You can control which parts are selected by dragging to the left or right while initially defining the volume:

- Drag to the right to select parts that are completely inside the volume only. The volume will have red edges when dragged to the right.

Note:



This action will not select sub-components of a monolithic JT in the viewer. The entire monolithic JT must be contained inside the volume to select it using the contained only mode.

- Drag to the left to select parts that are contained by or intersect with the volume. The volume will have blue edges when dragged to the left.

Note:

This action will select all components of a monolithic JT if any of the sub-components intersect the volume.

Once you have initialized your volume, you can right-click inside the volume to change the select mode using the context menu.

- Select **Contains Only**  to cause the volume to select parts that are completely inside the volume only.
- Select **Intersecting**  to cause the volume to select parts that are contained by or intersecting with the volume.

The parts which are in the selection volume are highlighted in a solid color.

4. Drag or pull the corners or edges of the frame of the volume to resize the volume as needed to highlight the parts you would like to select.

If you click in the empty area in the viewer, the highlighting is removed, but volume selection remains active.

5. Drag a face of the volume to translate parallel to that face. This will move the box around without needing to rotate or adjust its size.
6. To complete the volume selection, click **Accept**.
7. To cancel the volume selection, click **Cancel** or press the **Esc** key on your keyboard.

The number of parts that you can select with a volume select action is dependent on the value of the site-level preference set by your administrator. While performing a volume select operation, if you exceed the maximum set limit, the following actions take place:

- A warning informing you about the current number of parts that are included in the volume (but not selected because they exceeded the limit) is displayed.
- The **Volume Select** command stays highlighted and the viewer remains in the volume select mode.
- The selection which exceeded the limit is ignored.

Note:

The limit is considered only for a single Volume Select operation. This means that the maximum limit condition is applied only to your current selection action and does not apply to the total number of selected parts.

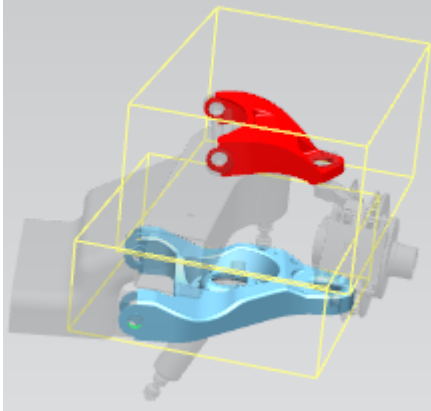
Display modes for models

You can choose to activate transparency or part color for parts in a model.

Based on what option you choose, the following behavior is observed for the selected parts.

Enable Transparent Selection mode is active

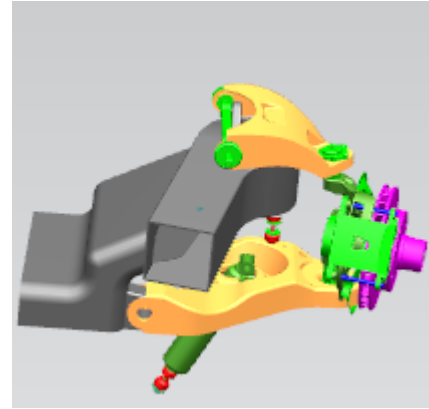
Selected parts are shown in their original color and with bounding boxes.



Unselected parts are shown as transparent. You cannot select transparent parts by clicking them in the viewer.

Part Color Selection mode is active

Selected parts are shown in the highlight color.



Unselected parts are shown in their original colors. You can select any displayed part by clicking it in the viewer.

Additionally, the behavior will differ in the client-side rendering mode. —

Enable Transparent Selection mode is active

The part or feature that you pre-highlight by hovering over it is highlighted (appears solid). All the other parts in the model remain transparent.

Only the parts that you select for measurement or getting geometrical information or the features which you select are highlighted (appear solid). All the other parts in the model remain transparent.

If you select a preexisting measurement or a geometric information box in the viewer, the measurement or the geometric information box and the parts being measured or for which you are getting geometric information are highlighted (appear solid). All the other parts in the model remain transparent.

Part Color Selection mode is active

The part or feature that you pre-highlight by hovering over it is highlighted. All the other parts in the model retain their original color.



The selected parts or part features in the model are highlighted. The remaining parts of the model retain their original color.

The selected preexisting measurement or a geometric information box are highlighted. The remaining parts of the model retain their original color.

Switch selection display mode in the 3D viewer

Perform the following steps to switch the selection display mode for a model:




1. **Load the product assembly** in the 3D viewer in Active Workspace.

2. Select the parts for which you wish to switch the selection display mode.
3. In the 3D viewer, from the work area toolbar, click **Transparent Selection**  or **Part Color Selection** , depending on which mode is currently enabled.

Based on the mode, the selected parts will either be shown in their **original color or in the highlight color**. Additionally, the **behavior will differ in the client-side rendering mode**.

Explode a model in the viewer

You can explode a model in the 3D viewer to enable you to see how the different parts comprising an assembly fit together and to better understand the spatial relationships between parts.

1. **Open the assembly** in the 3D viewer in Active Workspace.
2. From the work area toolbar, click **Explode** . If you don't find it on the work area toolbar, click **More Commands**  and select **Explode**  from the dropdown.

Note:

The *Explode* feature is available only in the client-side rendering mode. Additionally, it is only available in the structure viewer, and not in the part or universal viewer.

The **Explode** slider is displayed under the 3D viewer options ribbon.



3. Drag the slider or click the + or - button to adjust the level of explosion you wish to have for the model or assembly.



Dragging the slider explodes all parts, whether visible or not. Additionally, *explode* is based on center of the model, and not based on just the visible parts of the model.

4. To reassemble the model, click **Reset** or close the exploded view.

Note the following points:

- Selecting any other feature on the work area toolbar resets the exploded view.
- Siemens Digital Industries Software recommends clearing any sections or measurements prior to working with exploded views. Measurements and section planes will remain stationary as parts are moved to their exploded positions and could clutter the view of an exploded assembly.

Tip:





After you have exploded the parts, you can capture them in context with the 3D data within snapshots using the **Capture Snapshot**  command. All snapshots are saved in the **Gallery** .

Pack or unpack structure elements

You can group similar elements in an assembly by *packing* them together in a single level of the assembly. This results in a shorter and simpler structure display. The packed elements are represented using a single row in the structure with a numeric value appended to this row to indicate the number of packed items.

You can individually pack or unpack the selected elements in the structure that meet the packing criteria. You can also pack or unpack all elements in a selected structure simultaneously with a single click.

When you select a packed node in the tree, all the part instances packed in that node are highlighted in the 3D viewer. Additionally, if you choose to use Active Workspace Application Connect to view the structure in Lifecycle Visualization, the structure is displayed in the unpacked state initially.

To pack elements	To unpack elements
<p>To pack the selected elements in the structure that meet the packing criteria:</p> <ol style="list-style-type: none"> 1. Select one of the elements. 2. Click the Expand  icon. 3. Select Pack. 	<p>To unpack the elements in a structure:</p> <ol style="list-style-type: none"> 1. Select the packed element. 2. Click the Expand  icon. 3. Select Unpack.
<p>To pack all the elements in the structure:</p> <ol style="list-style-type: none"> 1. Click the Expand  icon. 2. Select Pack All. 	<p>To unpack all the elements in the structure:</p> <ol style="list-style-type: none"> 1. Click the Expand  icon. 2. Select Unpack All.


Note:

While finding elements within a structure or filtering a structure (Smart Discovery-indexed), the state of the structure remains the same, regardless of whether the structure elements in it are packed or unpacked manually or by default. However, the search results on the **Find** panel are always displayed as unpacked.

You can specify the reference designator values in the International Organization for Standardization (ISO) and in the American Society of Mechanical Engineers (ASME) formats. In the ASME format, the reference designator value is in the capital letter followed by a number and suffix letter. For example, **S1A**.

View non-indexed information for a structure

When working with a product structure, by default, the 3D viewer loads the latest indexed data associated with it. If you make any changes to the structure that have not already been indexed, you can still load the non-indexed information associated with the structure in the 3D viewer.

1. **Load the structure** in the 3D viewer.
2. On the work area toolbar, click **Viewer Options** .
3. In the **ModelType** section, clear the **Use indexing information** check box.

Note:

You can see the **ModelType** section only if model in the viewer is indexed.

10. Visualize photorealistic digital twin

Overview of visualizing your photorealistic digital twin


The Teamcenter lifecycle visualization Digital Reality viewer is a browser-based application integrated into Active Workspace. It enables you to view large scale, high quality, photorealistic digital twin visualizations. By leveraging the NVIDIA Omniverse platform within Teamcenter, the Digital Reality viewer supports advanced visualization capabilities using the Universal Scene Description (USD) format.

With the **Share with Industrial Metaverse** command used by the NVIDIA Omniverse platform, you can translate your digital twin JT representation, including the product structure and 3D information, to the USD format on-demand. This USD file represents the photorealistic digital twin visualization, which can then be rendered in the Digital Reality viewer.

Sharing with the Industrial Metaverse

You can use Active Workspace lifecycle visualization Digital Reality viewer to visualize your photorealistic digital twin and visual environments using NVIDIA Omniverse platform in Teamcenter.

Procedure

1. **Open your model** in the 3D viewer.
2. Click **More Commands ...** > **Share**  > **Share with Industrial Metaverse**.

If this is your first time in a session using the command, a window will appear prompting you to enter your credentials. Once the authentication is complete, the conversion and upload process will begin. A pop-up message will appear when the conversion is done.

Note:


There is a delay between the message appearing and the ability to visualize the model in the Digital Reality viewer. This delay is caused by the process that occurs after the conversion from JT to USD, where the resulting USD model is uploaded to the Nucleus database storage. With larger models, this process can take a couple of minutes.

3. Once the processing is complete, from the work area toolbar, choose **Details** > **XR** and review the rendering of the model.

Changing the visual environment

You can change the visual environment to enhance your ability to visualize and interact with your digital twin in various settings.

Procedure

1. **Open your model and choose Details > XR.**
2. From the work area toolbar of the Digital Reality viewer, click **Scene Gallery** .
3. From the **Gallery** panel, select a scene to visualize your photorealistic digital twin in a different visual environment.

11. Positioning parts and assemblies

Overview of positioning parts and assemblies

Positioning parts (and assemblies) is moving parts with respect to other parts in the viewer and also includes rotating parts and subassemblies of a 3D model. In Active Workspace, you can translate and rotate parts but you cannot scale them. Additionally, you can translate parts only in the client-side rendering mode.

Note:

You can move parts only if you have either a Professional or a Mockup Visualization service level.

You can perform the following actions:

- **Translate parts by dragging** them in the 3D viewer.
- Translate and rotate parts by using **part manipulators**.
- Translate and **rotate parts to align them** with other parts.

Using a screen, part, or global coordinate system, you can translate your parts along the X, Y, or Z axis.



Tip:

After you complete the movement or repositioning of parts, you can **capture them in a snapshot** or in a **session**. Additionally, you can save the position of the moved parts in the Teamcenter BOM.

Drag parts and assemblies in the 3D viewer


In Active Workspace, you can translate parts by dragging them in the 3D viewer. You translate parts using the screen coordinate system. You can also reset the parts to their original positions.


Procedure

1. **Open an assembly** in the 3D viewer.
2. On the work area toolbar, click **Move** . If you don't find it on the work area toolbar, click **More Commands**  and select **Move**  from the dropdown.

Note:

The move parts feature is available only if you have either a Professional or Mockup Visualization service level. Additionally, it is only available in the client-side rendering mode.


3. Select parts that you want to move either by clicking them in the 3D viewer, or in the tree.
4. On the secondary toolbar, click **Drag** .

Alternatively, you could click **Drag**  first and then click the part that you wish to move.

5. In the 3D viewer, you may want to use the navigation (Nav) cube to change the view orientation or screen space, and then drag the selected part to its desired position in the current orientation.
 - a. Click the side handles of the Nav cube to select the orientation along the axis that you want.
 - b. Click the corners and edges of the Nav cube to get isometric views.

Note:

The dragged parts retain their position in the viewer if you close the secondary toolbar. However, closing the secondary toolbar disables the **Drag** command.

6. (Optional) Click **Save Position**  on the secondary toolbar if you wish to save the transformation position of the dragged parts to the Teamcenter BOM.

Note:

Only those parts to which you have *write* access are saved.




Use manipulators to move selected parts and assemblies

Interactive part manipulators are one of several tools you can use to move 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.

You can do the following with manipulators to **move parts**:


- Translate along and rotate about the axis.
- Enter delta angle or distance values.
- Enter X, Y, Z coordinates.

Procedure

1. **Open an assembly** in the 3D viewer.
2. On the work area toolbar, click **Move** . If you don't find it on the work area toolbar, click **More Commands**  and select **Move**  from the dropdown.

Note:

The move parts feature is available only if you have either a Professional or Mockup Visualization service level. Additionally, it is only available in the client-side rendering mode.

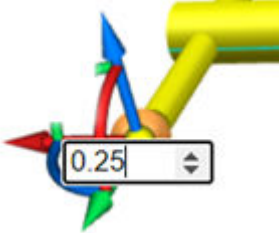
3. Select parts that you want to move either by clicking them in the 3D viewer, or in the tree.
4. On the secondary toolbar, click **Manipulate** .

A manipulator appears at the center of the bounding box of the selected part. The manipulator is an interactive triad with three axis arrows for translating and three arcs for rotating.

The manipulator retains its position in the viewer even if you close the secondary toolbar and can be used to move parts.

5. Transform the parts in any of these ways.

To do this	Perform this action
Drag the parts in X, Y, or Z direction.	Drag the X (red), Y (green), or Z (blue) manipulator axis.
Rotate parts around an axis.	Drag one of the arcs of the manipulator.
Translate or rotate parts using precise values in X, Y, or Z coordinates.	<ol style="list-style-type: none"> Right-click the manipulator sphere. The Manipulator Align dialog appears. You can move the dialog box to an appropriate location on the screen. Select the coordinate system of your choice: <ul style="list-style-type: none"> • Part Coordinate System • Part Bounding Box Center • Global Coordinate System Select Show Position. Use either of these options:


To do this	Perform this action
	<ul style="list-style-type: none"> • In the Position and Rotation dialog, do the following based on your requirement: <ul style="list-style-type: none"> A. Enter the precise distance for moving the selected part along the X, Y, or Z axis. The default unit of movement is meters. B. Enter the precise rotation value for rotating the selected part along the XY, YZ, or XZ plane. • Drag the X, Y, or Z axis of the manipulator sphere as required to the desired position or click the desired arc and rotate it to the required position. • Click and slightly drag the X, Y, or Z axis of the manipulator sphere as required and enter the precise distance in the value box that appears for moving the selected part along the X, Y, or Z axis to the desired position.  <p>Click and slightly rotate the desired arc and enter precise rotation value for rotating the selected part along the XY, YZ, or XZ plane.</p> <p>The part is moved to the desired position based along the coordinate system you chose earlier.</p>
Specify the coordinate system to translate parts.	<ol style="list-style-type: none"> a. Right-click the manipulator sphere. The Manipulator Align dialog appears. You can move the dialog box to an appropriate location on the screen. b. Select Move Only Manipulator. This moves only the manipulator while the part remains stationary in the viewer. c. Select one of the following: <ul style="list-style-type: none"> • Part Coordinate System • Part Bounding Box Center

To do this	Perform this action
	<ul style="list-style-type: none"> • Global Coordinate System
Snap to feature geometry.	<p>a. Right-click the manipulator sphere.</p> <p>The Manipulator Align dialog appears. You can move the dialog box to an appropriate location on the screen.</p> <p>b. Select Snap To Part or Feature.</p> <p>c. Select the part feature (surface, edge, or point) to which you wish to snap the transformed part.</p> <p>d. Double-click the desired arc (for rotating 180°) or click the coordinate arrows (to flip the part) to ensure that the transformed part aligns with its new position as required.</p>
Rotate the parts by aligning an axis of the manipulator to an edge, vertex, or a surface.	<p>a. Right-click the manipulator sphere.</p> <p>The Manipulator Align dialog appears. You can move the dialog box to an appropriate location on the screen.</p> <p>b. Select Align To Axis.</p>

Align parts and assemblies with other parts


You can align parts with other parts by selecting a feature on one part to align with a feature on another part. For example, you can align an edge on one part with an edge on another part.

Procedure

1. **Open the model** in the 3D viewer.
2. In the 3D viewer, click **Move**  on the toolbar.

Note:


The move parts feature is available only if you have either a Professional or Mockup Visualization service level. Additionally, it is only available in the client-side rendering mode.

3. Select the parts that you want to move either by clicking them in the 3D viewer or in the tree.
4. Click **Align**  on the toolbar.
5. From the selected parts in your 3D viewer, select a geometric *source* feature such as an edge or a surface. You can use the feature filter buttons on the toolbar to control which types of features are

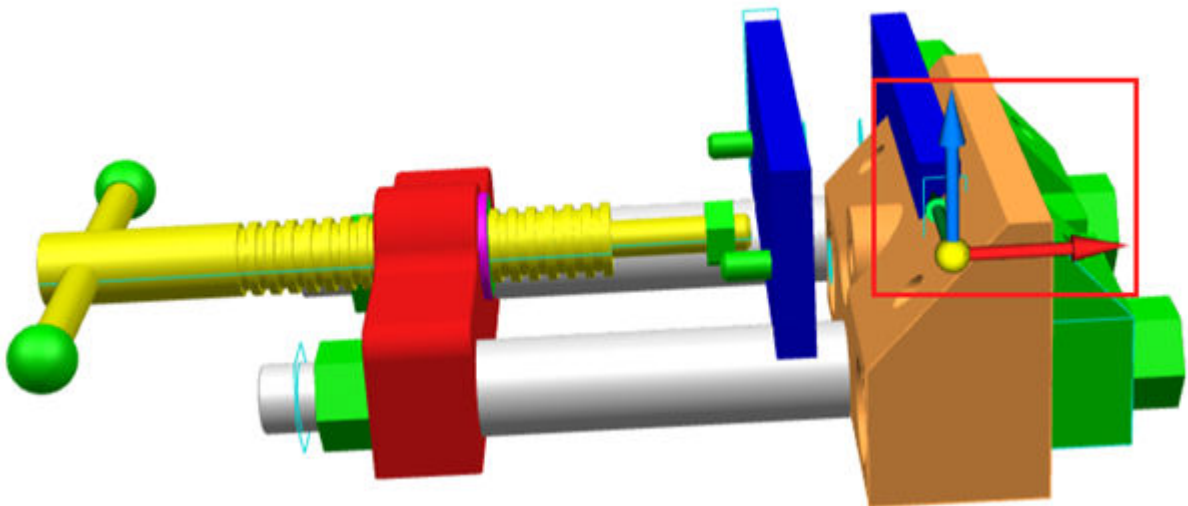
selectable. This *source* feature is translated and rotated to align with the next feature you select, which is the *target* feature. This same translation and rotation is applied to all the selected parts.

Tip:


To exit the alignment operation before selecting the target feature, press the Esc key on your keyboard.

6. Select a different target feature to align with the source feature. All the selected parts are subsequently translated and rotated such that the source and target features align.
7. (Optional) You can manually adjust the alignment using a manipulator. For example, you can rotate the moved parts by 180° to achieve the desired alignment. To do this:
 - a. Click **Flip** .

A manipulator appears.



Note:

The **Flip**  command is enabled only after you select the source and target alignment features.

- b. To flip the selected parts around the aligned features by 180°, double-click the manipulator axis normal to the alignment plane you wish to flip the selected parts around.
- c. The manipulator can also be used to manually translate or rotate the selected parts.






Reset parts and assemblies to their original position

After you have completed visualizing parts in their moved state, you can reset either any specific parts of the model that you had moved earlier, or all moved parts back to their original position.

Note:

You can reset only those parts that you have **moved** earlier.

Procedure

1. In the 3D viewer, from the work area toolbar, click **Move** . If you don't find it on the work area toolbar, click **More Commands**  and select **Move**  from the dropdown.
2. On the secondary toolbar that appears, perform either of these actions:
 - To move back only specific parts to their original position, select the parts, and click **Reset** .
 - To move back all parts to their original position, click **Reset All** .


12. Focusing on components



Hide or display components

You can turn off and turn on the display of assembly components (parts). You can set visibility in the following ways:


- In a component list, click the icon for a component to switch its visibility on or off. This way, you can choose to view specific parts by selecting items from the tree.

You can switch the visibility status on and off regardless of the selection status.

- On the toolbar, click **Visibility Controls** , and based on your requirement, click the desired command from the list.

For example, if you wish to hide the selected components, click **Hide Selected** . Click **Show All**  in the list to view the entire model.

Note that the availability of commands depends on the current selection and visibility state.

- Choose the desired command for **parts or subassemblies you wish to see or hide** from the context menu in the 3D viewer.
- Use the **Viewer Options** panel to view or hide all components. The **Viewer Options** panel is available in both, the **Details** and the **3D** view.
 1. On the toolbar, click **Viewer Options** .
 2. In the **Viewer Options** panel that opens, under **Render Options**, select **Show All Components on Load** to see all components when the assembly loads in the 3D viewer.

Deselect **Show All Components on Load** to hide all components when the assembly loads in the 3D viewer.

Tip:

Access the **Viewer Options** panel from the **Details view** before loading the model in the 3D viewer, which can be helpful in case of large assemblies.

You can also **search for and filter components** by volume or by proximity to target parts. Visibility of matching parts is turned on, while that of non-matching parts is turned off.

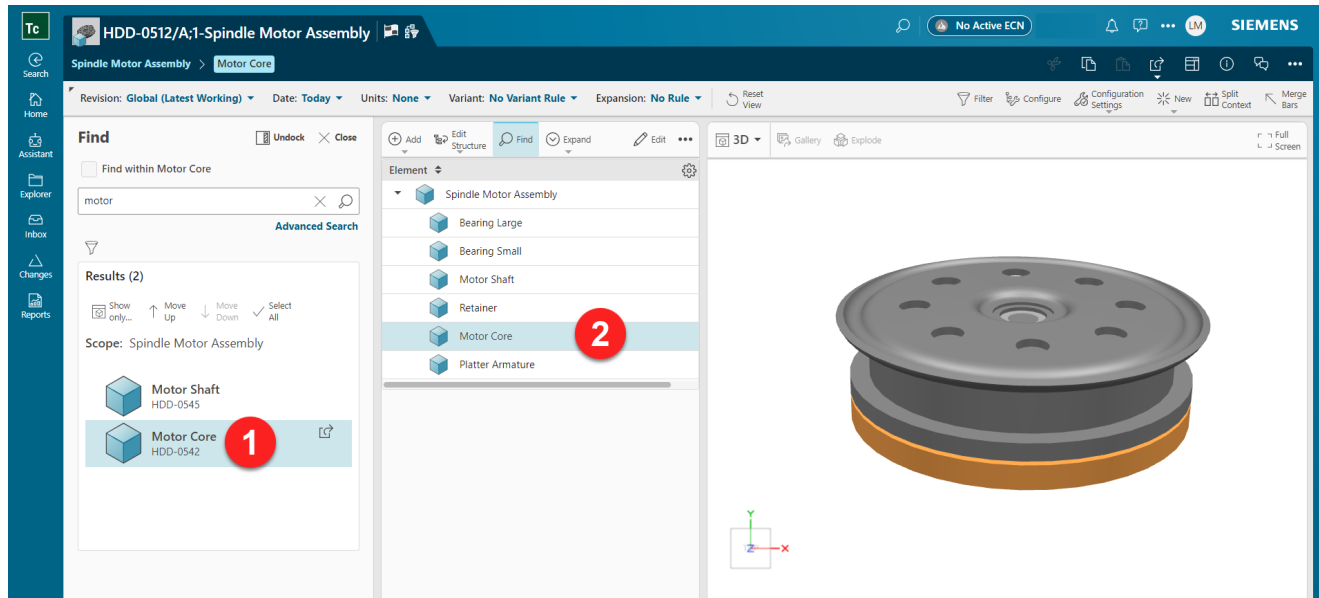
Note:

Your administrator can disable the viewer from loading the model when the model is opened. This is useful when loading large assemblies.

Select components

You can select a component in either of the following ways:

- When you know where to find the component in the model, you can select it in the viewer.
- When you know the component name, you can select it by clicking the component name in a list such as:
 - The results of an in-context search
 - An active structure list




To make this selection	Do this
Select a component	In either the viewer or in a list, click or tap the component.
Select all components contained by the parent assembly of a selected component	In the viewer, click or tap a selected component.

To make this selection	Do this
Select an additional component	<p>When the multiselect mode is on:</p> <p>With a mouse</p> <ul style="list-style-type: none"> • In the tree view, click the square against the desired component. • In the 3D viewer, hold Ctrl and click the component. <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Note: Multiselecting in the 3D viewer does not select the corresponding elements in the Tree with Summary view.</p> </div> <p>On a touch device</p> <ul style="list-style-type: none"> • Press and hold the component. • While multiselect mode is turned on, you can click or tap additional components to add them to the selection. <p>When the multiselect mode is off, to add a component to the earlier selection, in the 3D viewer, hold Ctrl and click the component.</p>
Select all components	<ol style="list-style-type: none"> 1. Turn on the multiselect mode. <ul style="list-style-type: none"> With a mouse In the 3D viewer, hold Ctrl and click a component. Alternatively, in the tree view, click the square against the desired component. On a touch device Press and hold a component. 2. From the work area toolbar, click Select All ✓.
Deselect all components	<p>Use either of the following options:</p> <ul style="list-style-type: none"> • Click or tap the viewer background. • Click or tap the root of the breadcrumb.

Adjust the visibility of components

You can adjust the visibility of components based on the current selection using any of the following methods:

Choose from the Visibility Controls commands

The visibility commands, such as **Show All** appear in the **Visibility Controls**  list on the toolbar, as applicable for the current visibility status and selection.


Select this command	To do this
Show All	Turn on the visibility of all components.
Hide All	Turn off the visibility of all components.
Show Selected Only	Turn on the visibility of the currently selected components, and turn off the visibility of unselected components.
Hide Selected	Turn off the visibility of the selected components.
Show Selected	Turn on the visibility of the selected components.
Clean 3D View	Remove content such as measurements and sections, reset part positions and colors, and hide PMI.

Choose the desired command from the context menu

You can right-click in the 3D viewer and choose the desired command from the context menu.

Use the Viewer Options panel to set viewing or hiding all components

You can use this option only if you wish to either see or hide all components when the assembly loads in the 3D viewer.

1. On the toolbar, click **Viewer Options** .
2. In the **Viewer Options** panel that opens, under **Render Options**, select **Show All Components on Load** to see all components when the assembly loads in the 3D viewer.

Deselect **Show All Components on Load** to hide all components when the assembly loads in the 3D viewer.

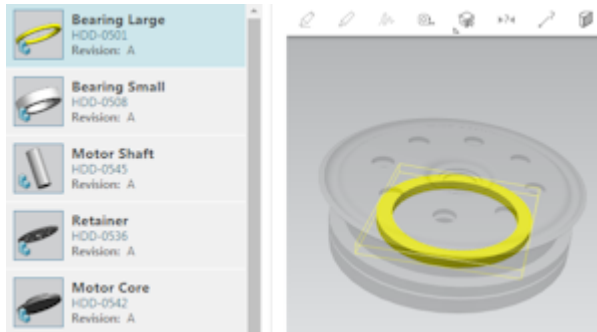
Selection status examples

Selection status and visibility settings affect the display of active content.

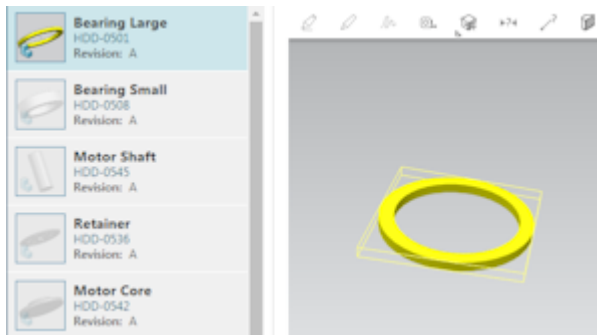
In the following display examples, **Transparent Selection**  is selected.



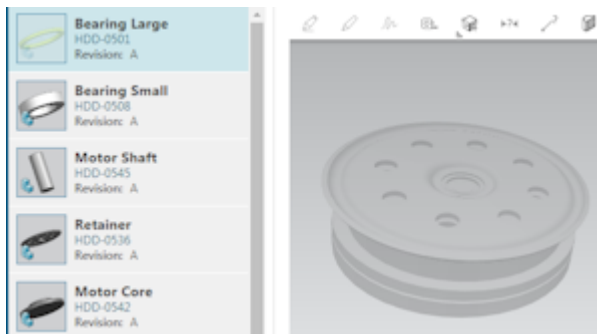
Nothing is selected.
All components are turned on.
Components have their original coloring.



The **Bearing Large** component is selected.
All components are turned on.
All content except the selected component is transparent.



The **Bearing Large** component is selected.
All unselected content is turned off.



The **Bearing Large** component is selected and turned off.
All unselected components are turned on and transparent.

13. Searching for and filtering parts

Search and filter by volume



You can choose parts as targets and perform spatial volume searches against them. Each time you add or delete a target part, the corner values change to represent the smallest bounding box that can contain all the selected parts. You can manually change the bounding box corner values; however, manual changes are overwritten when you add or remove a part from the list of targets.

The units for the corner values are based on the **Displayed** units setting available on the **Viewer Options** panel. The default display units are **Meters**.

Your administrator can configure different default units. Use the **Displayed** units setting to **override the default units**.

Note:

The search and filter by volume feature is only available with the Mockup Visualization service level.

1. **Open an assembly** in the 3D viewer.
2. On the 3D viewer, from the work area toolbar, click **More Commands ... > Volume** .
3. In the **Volume** panel, choose target parts to define the volume.
 - a. Select a component, either from a component list on the left or by clicking a component in the 3D viewer.
 - b. Click **Add Target** .

Note:

Add Target is enabled only after you select a component (that is not already in the target list) either in the component list (primary work area) or in the viewer. Additionally, the targets that you add are retained for the duration of your session, even after you close the panel.

- c. Under **Target Volume Coordinates**, specify or edit the minimum and maximum values for the X, Y and Z coordinates. Alternatively, you can do one of the following to update the values.
 - Drag or pull the corners or edges of the frame to resize the volume.
 - Drag a face of the volume to translate the box parallel to that face.

Note:

Target Volume Coordinates is enabled and initialized after a component is added.

- d. Choose the **Volume Boundary** filter option.
 - To filter only parts that are completely inside the volume, choose **Contains Only**.
 - To filter parts that are inside or intersect the volume, choose **Intersecting**.

By default, it is set to **Intersecting**.
- e. Choose one of the following **Actions** to filter results by volume.
 - **Show Only** (default): The filter turns on all target parts and any parts that intersect or share volume within the bounding boxes of the target parts. All other parts are turned off in the viewer.
 - **Select**: The filter selects only those parts whose geometries intersect with the volume of the selected part without impacting the visibility of parts that do not pass the filter.
- f. To add more target parts, repeat the preceding steps.

When you add a part to or remove a part from the target list, the minimum and maximum corner values are updated to reflect the volume of the target list.

4. Click **Apply**.


Search and filter by proximity


You can choose target parts and perform proximity searches against them. Valid values for proximity distance are greater than or equal to zero.

The units for the proximity distance value are based on the **Displayed** units setting available on the **Viewer Options** panel. The default display units are **Meters**. Use the **Displayed** units setting to **override the default units**.

Note:

The search and filter by proximity feature is only available with the Mockup Visualization service level.

1. **Open an assembly** in the 3D viewer.
2. From the work area toolbar, click **Proximity** .

3. In the **Proximity** panel, choose target parts to define the volume.
 - a. Select one or more components, either from a component list on the left or by clicking a component in the 3D viewer.
 - b. Click **Add Target** .

Note:

Add Target is enabled only after you select a component (that is not already in the target list) either in the component list or in the viewer. Additionally, the targets that you add are retained for the duration of your session, even after you close the panel.

- c. To add more target parts, repeat the preceding steps.
4. In the **Filter visibility to display parts only within the specified distance in (units)** box, enter the proximity value.

Note:


Valid values are greater than or equal to zero.


5. Click **Filter**.

The search makes visible all target parts and any part within the specified distance of the target parts. All other parts are not visible in the viewer.

Search by name and show only the results


You can search by name for occurrences within an open indexed structure, and then display only the search results in the viewer.

1. **Open an assembly** that has been indexed in the database.
2. From the results panel toolbar, click **Find** .


If the **Find** command is not visible on the results panel toolbar, click **More Commands**  to display additional commands that are not displayed on the toolbar due to size constraints.

Note:

The **Find** command is available for models that have been indexed.

3. In the **Find** panel search box, enter a name and then click **Search** .

Occurrences that match the search pattern are displayed in the **Results** list.


4. (Optional) To narrow the results, click **Filters** and enter a value for one or more attributes.
5. To display only the components listed in the search results in the viewer, click **Show only results in viewer** .

Visibility of all results is turned on, and visibility of all nonmatching occurrences is turned off.

14. Measuring 3D models

Change the working unit of the viewer

Based on your requirement, you can change the working unit of the Active Workspace 3D viewer. The selected unit becomes the default unit for performing persisted measurements, sections, or volume and proximity searches. Linear measurement values shown in the **Measure** panel are converted to the displayed unit for new measurements. However, existing measurements are based on the **Displayed Units** unit setting at the time the measurement was taken.

1. **Open your model** in the 3D viewer.
2. From the work area toolbar, click **Viewer Options** .
3. Under **Units**, from the **Displayed Units** list, select a new linear unit value.

Note:

The value that is displayed in the **Model Units** field is based on a registry preference where the Visualization server is located and is currently hard coded to *Meters*.

Taking measurements in 3D models

For 3D models, you can take measurements between:

- Features, including surfaces, edges, vertices, points, and arc centers
- Parts
- A part and a feature

Measurement data gets auto-saved during a working session. So, if you return to a previous working session, you have the option to restore that information or continue working with the product structure in its default state.

The default display unit is **Meters**. You can use the **Displayed Units** setting to **override the default unit**. Linear measurement values shown in the **Measure** panel are converted to the displayed unit for new measurements. However, existing measurements are based on the **Displayed Units** unit setting at the time the measurement was taken.



As per the convention:

- Measurements to a circular edge are made to the center of the circle.


- Measurements to a straight edge (line) are the shortest distance to the infinite line containing the edge.
- Measurements to a planar face are the shortest distance to the infinite plane containing the face.
- Measurements to a cylindrical or conical face are the shortest distance 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.
- Measurements of two straight edges or cylindrical axes either display the minimum distance if they are parallel or skewed or the angle if they intersect.
- Measurements of two straight edges or cylindrical axes that are not parallel and do not intersect (are not coplanar) display both the minimum distance and the angle.

Measure and add measurement data in 3D models

For 3D models, **you can take measurements** between parts, features of parts, and a part and a feature.

1. **Open the product or assembly** in the 3D viewer.
2. On the toolbar, click **Measure** . If you don't find it on the toolbar, click **More Commands ... >** **Measure** .
 - The secondary toolbar opens, and the list of features, including surfaces, edges, points, vertices, and arc centers is selected.

Tip:
To deselect a particular feature, click on it.

 - To create a measurement between parts, select **Parts** . The feature group gets deselected.

Note:
Until you log out of the current session, the secondary toolbar retains the last selected filter (feature or part) and makes it the selection when you access it again.
3. Click the first part or feature, and then click the second part or feature.

The measurement calculation and data returned depends on the parts or features you pick.

Note:

You can stop the creation of a measurement after selecting the first element and before selecting the second element by pressing the Esc key on your keyboard. Pressing the Esc key at any other time with the **Measurement** toolbar active will hide the toolbar.

The following behavior is observed when the client-side rendering mode is the active mode.

When transparency is enabled for a model

If you pre-highlight a part or a pre-existing measurement by hovering over it, the part or the measurement box is highlighted. It appears solid while the remaining parts appear transparent.

If you select parts for measurement, the selected parts and the measurement box are highlighted (appear solid). All remaining parts in the model appear transparent.

If you select a preexisting measurement in the viewer, the measurement box and the measured parts are highlighted (appear solid). All the other parts of the model appear transparent.

When transparency is disabled for a model

If you pre-highlight a part or a pre-existing measurement by hovering over it, the part or the measurement box is highlighted, while the remaining parts of the model are displayed in their original color.

If you select parts for measurement, the selected parts are highlighted in the selection color. The remaining parts of the model retain their original color.


If you select a preexisting measurement in the viewer, the measurement box and the measured parts are highlighted. The remaining parts of the model retain their original color.

If the model or assembly contains incorrect or broken edge information, based on the currently active rendering mode, the following actions occur.

Rendering method**Action that happens**


Client-side rendering (CSR) An error message is displayed, and measurement is not calculated or displayed.



Server-side rendering (SSR) Depending on the parts or features you pick, the measurement calculation and data is returned.

4. To copy the measurement data, select the measurement text box and on the secondary toolbar, click **Copy Text** .
5. Open any text editor and paste the copied text.

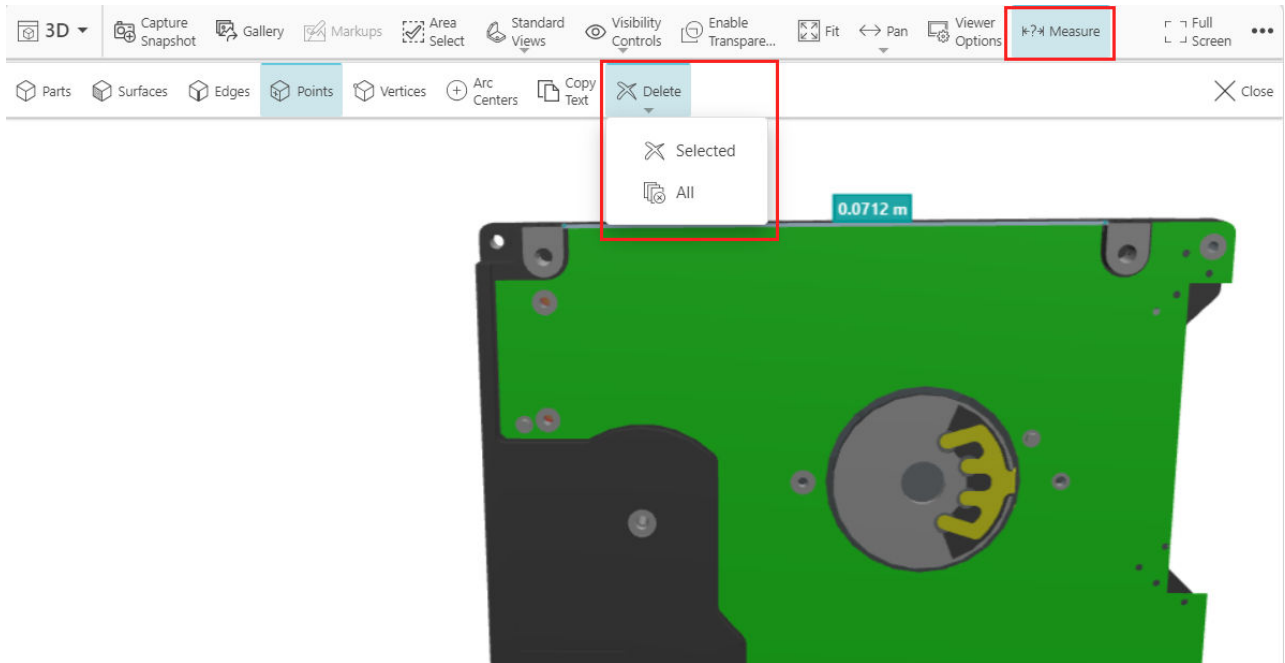
Delete measurements





Perform the following steps to delete measurements added to an already **opened assembly** in the Active Workspace viewer:

1. **Open your model** or assembly in the 3D viewer.
2. From the work area toolbar, click **Measure** .

If you do not find **Measure** on the work area toolbar, click **More Commands** , and select **Measure** .

3. On the secondary toolbar, select the appropriate option to delete a single or all measurements.



To delete a single measurement	To delete all measurements
<ol style="list-style-type: none"> Select the measurement annotation on the model. On the secondary toolbar, click Delete . From the list, click Selected . <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note:</p> <p>Alternatively, with or without the Measure toolbar active, you can either right-click a selected measurement and select Delete from the context menu, or press Delete on your keyboard.</p> </div>	<ol style="list-style-type: none"> On the secondary toolbar, click Delete . From the list, click All . In the confirmation message box that appears, click Delete.

4. If you have restored measurements, you cannot remove those using the steps listed earlier. You can only remove them using the **Clean 3D View** command.

From the work area toolbar, select **Visibility Controls**  > **Clean 3D View** . Alternatively, right-click in the viewer, and select **Clean 3D View**  from the context menu.

Associative measurements are 3D measurements that remember the objects for which it provides measurements. XTRep data or JTRep data translated with CAD tags is required for creating associative measurements. *Restored measurements* are added to labels of non-associative measurements to indicate that the information may be out of sync with the 3D data.




Caution:

Clean 3D View also removes all the **user modifications** such as measurements, geometric information, and sections, resets part positions and colors, and hides PMI.

15. Managing geometric information for 3D models

Generate geometric information for a part or feature in a 3D model

You can get geometric data about entities (points, vertices, edges, faces or surfaces, and arc centers) of parts in an assembly. Perform the following steps to create geometric information for entire parts or for features of parts, such as surfaces or edges:

1. **Open your model** or assembly in the 3D viewer.
2. From the work area toolbar, click **Geometric Information** . If you don't find it on the work area toolbar, click **More Commands**  and select **Geometric Information**  from the list.

The secondary toolbar opens displaying the selection filters for parts (to enable selecting entire parts) and for the individual geometric features of parts, such as surfaces, edges, points, vertices, and arc centers (to enable selecting individual geometric features).

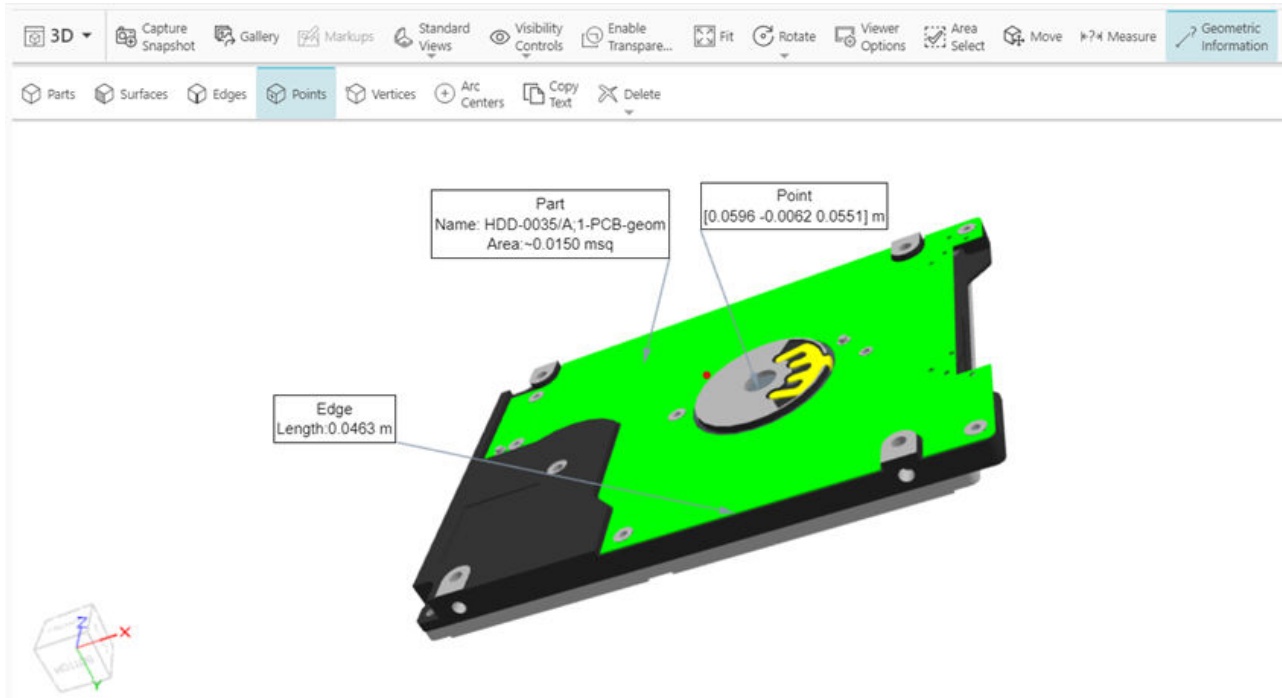
You can set the desired selection filter by toggling on either parts, or one or more of the geometric features.

Note:

When **Parts** is enabled, only entire parts can be selected. However, when any of the geometric feature types is enabled, **Parts** is disabled automatically.

3. Click a part or feature on your model or assembly, or a part entity, such as a surface or a point for which you wish to get geometric data.

Geometric information for that part or feature is annotated on your model. Additionally, the data is auto-saved during a working session. So, if you return to a previous working session, you have the option to restore that information or continue working with the default state of the product structure.

**Note:**

Pressing the Esc key with the **Geometric Information** toolbar active will hide the toolbar.

The following behavior is observed when the client-side rendering mode is the active mode.

When transparency is enabled for a model	When transparency is disabled for a model
<p>If you pre-highlight a part, a feature, or a preexisting geometric information box by hovering over it, the part, feature, or the geometric information box are highlighted (appear solid). All remaining parts of the model appear transparent.</p> <p>If you select parts or features for getting geometric information, the selected parts or features and the geometric information box are highlighted (appear solid). All remaining parts in the model appear transparent.</p> <p>If you select a preexisting geometric information box in the viewer, the geometric information box and the parts or features for which you are getting geometric information are highlighted (appear solid). All the other parts of the model appear transparent.</p>	<p>If you pre-highlight a part, a feature, or a preexisting geometric information box by hovering over it, the part, feature, or the geometric information box are highlighted. The remaining parts of the model retain their original color.</p> <p>If you select parts or features for getting geometric information, the selected parts or features, and the geometric information box are highlighted in the selection color. The remaining parts of the model retain their original color. In short, the selection of parts is not affected by transparency.</p> <p>If you select a preexisting geometric information box in the viewer, the geometric information box and the parts or features for which you are getting geometric information are highlighted. The remaining parts of the model retain their original color.</p>

When transparency is enabled for a model	When transparency is disabled for a model
--	---


	color. In short, the selection of parts is not affected by transparency.
--	--

If the model or assembly contains incorrect or broken edge information, based on the currently active rendering mode, the following actions occur.

Rendering method	Action that happens
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Client-side rendering (CSR)	An error message is displayed and geometric data is not displayed.
-----------------------------	--

Server-side rendering (SSR)	Depending on the entities of the parts or features you pick, geometric data is displayed.
-----------------------------	---

- To copy the geometric data, on the secondary toolbar, select the geometric information annotation and click **Copy Text** .


The data gets copied to the clipboard. To retrieve it, open any text editor and paste it.



Note:

The default display unit is **Meters**. Your administrator can configure a different default unit. Use the **Displayed** unit setting to **override the default unit**.

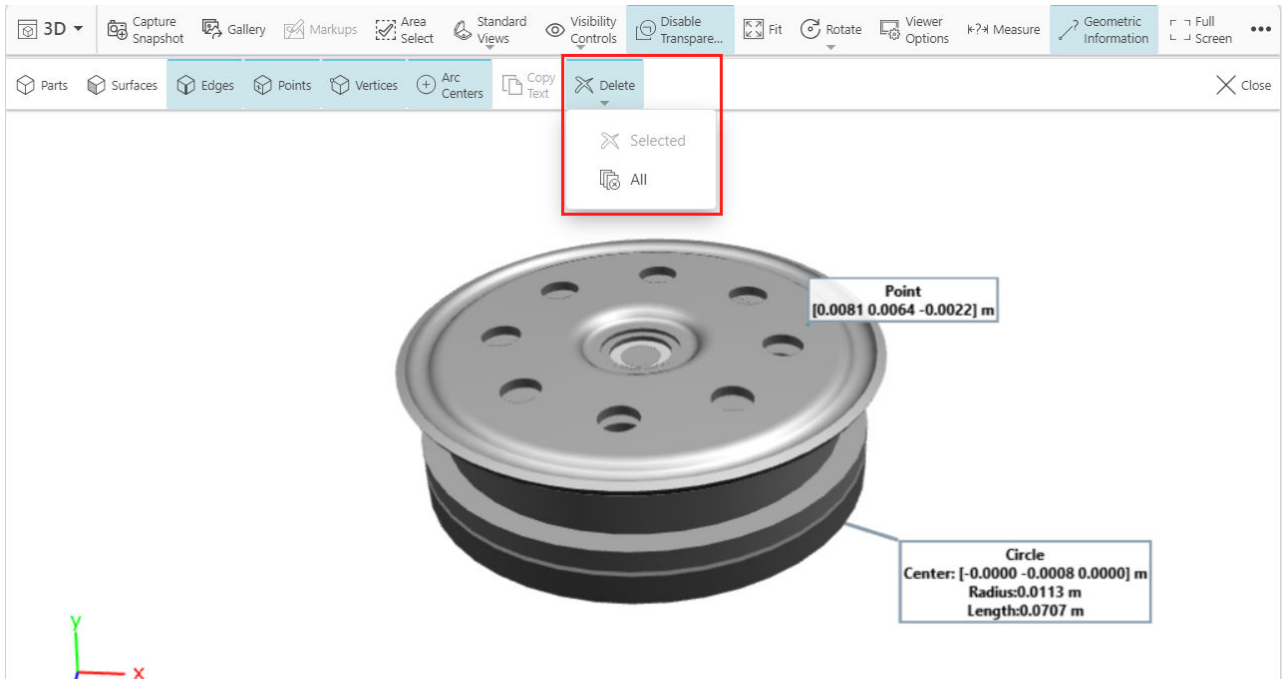
Delete geometric information annotations





Perform the following steps to delete geometric information annotations from an already **opened assembly** in the Active Workspace viewer:

- From the work area toolbar, click **Geometric Information** .

If you do not find **Geometric Information** on the work area toolbar, click **More Commands** , and select **Geometric Information** .

- Select the appropriate option to delete a single or all geometric information annotations.



To delete a single geometric information annotation	To delete all geometric information annotations
<ol style="list-style-type: none"> Select the geometric information annotation on the model. On the secondary toolbar, click Delete . From the list, click Selected . 	<ol style="list-style-type: none"> On the secondary toolbar, click Delete . From the list, click All . In the dialog box, click Delete.
<p>Note:</p> <p>Alternatively, with or without the Geometric Information toolbar active, you can either right-click a selected geometric information annotation and select Delete from the context menu, or press Delete on your keyboard.</p>	

- If you have restored geometric information annotations, you cannot remove those using the steps listed earlier. You can only remove them using the **Clean 3D View** command.

From the work area toolbar, click **Visibility Controls**  > **Clean 3D View** . Alternatively, right-click in the viewer, and select **Clean 3D View**  from the context menu.

Associative geometric information is 3D geometric information that remembers the objects for which it provides geometric information. XTRep data or JTRep data translated with CAD tags is required for creating associative geometric information. *Restored geometric information* is added

to labels of non-associative geometric information to indicate that the information may be out of sync with the 3D data.

Caution:

Clean 3D View also removes all the **user modifications** such as measurements, geometric information, and sections, resets part positions and colors, and hides PMI.


16. Sectioning 3D models

Create 3D sections

You can pass a plane through your model and display a cross section of the model where it is cut by the plane. The sections can be aligned along an axis or positioned perpendicular to an edge or on and parallel to a surface.

Note:

Cross sectioning persists between saved sessions and session information gets autosaved at appropriate times. So, if you create a cross section and log off without saving the session, the cross section information is still preserved within the session bookmark. You can also capture images of cross sections and apply markups to them.

1. **Load the assembly** in the 3D viewer.
2. From the work area toolbar, click **Create Section** .
3. Select the **XY**, **XZ**, or **YZ** axis option to create a section directly, or select **Normal to Edge** or **Coincident to Plane** option to create a section by then selecting an edge or a surface feature.

The option you choose is highlighted or enabled on the list. Click the option again to deselect it and choose another option.

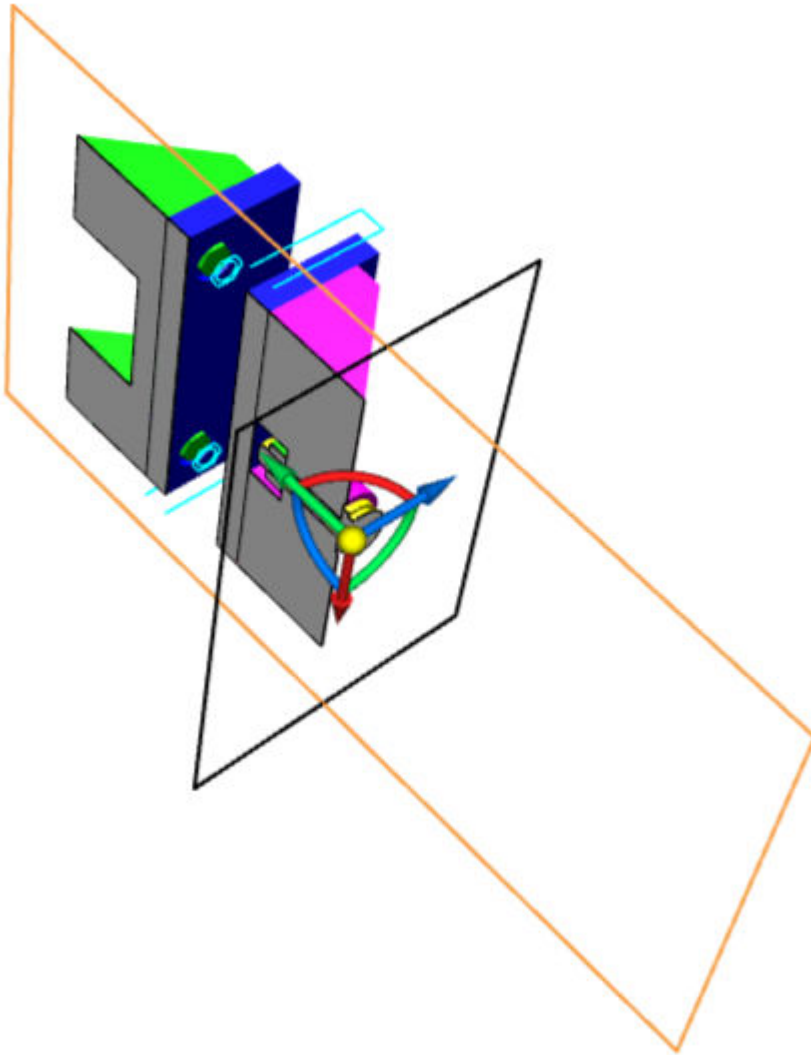
Note:

To abort section creation after selecting the **Normal to Edge** or **Coincident to Plane** options, press the Esc key on your keyboard before you click the edge or plane.


- A section is created and the **Section** panel is displayed. If the geometric feature information (STT data) for the model or assembly is missing or inaccurate, you may not be able to create a section using an edge or a surface. If this happens, an error message is displayed.
- A manipulator appears at the center of the bounding box of the section if you are in the client-side rendering mode. The manipulator is an interactive triad with three axis arrows for translating and three arcs for rotating.

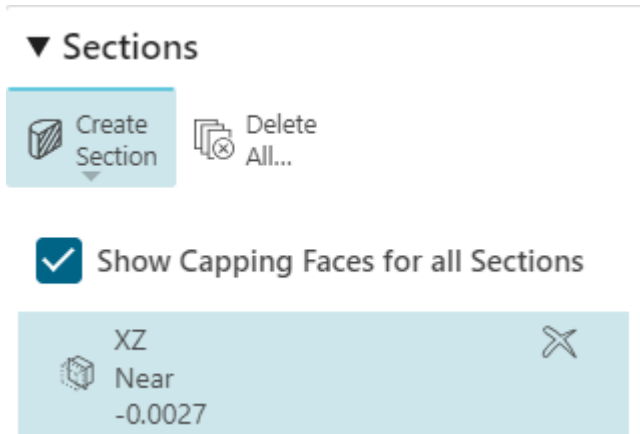
If the manipulator does not appear, right-click the section frame in the 3D viewer and from the context menu choose **Show Manipulator**. Repeat the same action if you wish to hide the manipulator.

- By default, the cut lines on the visible edges of the sectioned parts are not visible. To view the cut lines, in the **Section** panel, select **Edit Section > Show Cut Lines**.

**Note:**

The choice to either display or hide cut lines persists for the current and the future sections based on whether you select or remove the selection for **Show Cut Lines**.

4. To create additional sections, in the **Section** panel, under **Sections**, click **Create Section** . The new sections are added to the list.

**Note:**

You can only create a maximum of six sections in a user session.

The color of the selected section is the same as the part selection color of the **currently active scheme**. However, if you have created the section in standalone Lifecycle Visualization, and are now viewing it in Active Workspace, the section color from standalone Lifecycle Visualization is retained.

- (Optional) Select the **Show Capping Faces for all Sections** check box to cap the cut faces so that your models do not appear hollow.

Once the section is created, you can **edit** it to change its attributes such as its plane orientation, clipping direction, offset value, and visibility.

Edit 3D sections

You can edit section attributes, such as its plane orientation, clipping direction, offset value, and visibility. Provided you have preexisting sections, to do this, you can use any of these methods:

- **Use the Section panel**
- Select and pull the **section plane edge**
- Use the **section context menu**
- Use the **Translate to a Surface Point** option
- Use the 3D **manipulator**

Prerequisite

In a session, load an assembly and **create sections** on it, or restore a previous session containing sections.

Use the Section panel

1. On the toolbar, click **Section** . Note that **Section** is active only when there are existing sections.

The **Section** panel is displayed.

2. Under **Sections**, select the section that you wish to edit. The section is highlighted in the viewer.

Tip:

If the rendering mode is client-side rendering (CSR), you can also select the section by clicking it in the 3D viewer. This selects the section in the **Section** panel.

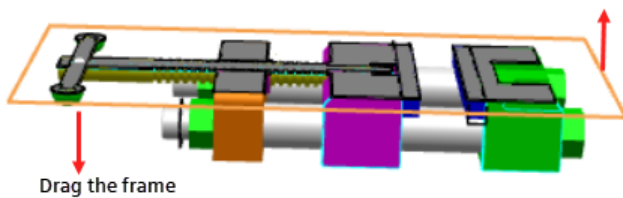
3. In **Edit Section**, perform the desired action.

To	Do this
Change the section plane alignment.	From the Orientation list, select an axis or a plane. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Note:</p> <p>If you are using a session file containing a section not aligned with the three principle axes, a custom value is displayed in the list. You can choose any of the other orientations, which changes the orientation plane accordingly. Choose the custom axis to return the plane to its original orientation.</p> </div>
Move the section plane along its axis.	Move the section plane to position it where it provides the best view of the internals of a product. <ul style="list-style-type: none"> • Under Offset, drag the slider or click the + or - button to adjust the value. • Enter the offset value manually in the Offset box. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Tip:</p> <p>To view the range of valid axis distance values, move the slider to the far left for the lowest value and to the far right for the highest value.</p> </div>
Show or hide cut lines for this specific section.	Select or clear the Show Cut Lines check box.

To	Do this
	<div style="border: 1px solid black; padding: 5px;"> <p>Tip:</p> <p>To improve performance, especially while manipulating the section plane location, clear the Show Cut Lines check box.</p> </div>
Set the clipping direction.	<p>The clipping direction value determines which side of the section plane is clipped, with respect to the current view.</p> <p>In the Clipping Direction list, select an appropriate value.</p>
Show or hide a section in the 3D view.	<p>Select or clear the Toggle Section Visibility check box.</p>

Use the section plane edge

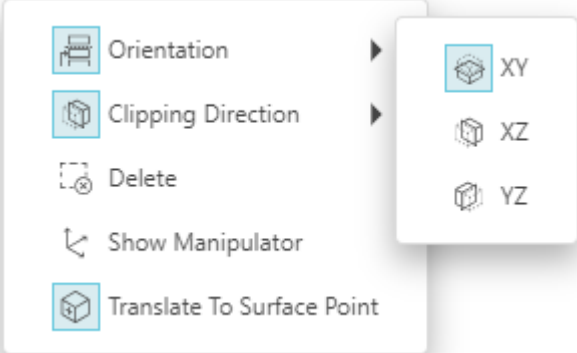

In the **3D** viewer, select and drag or pull the section plane frame (box) as required.



Use the section context menu

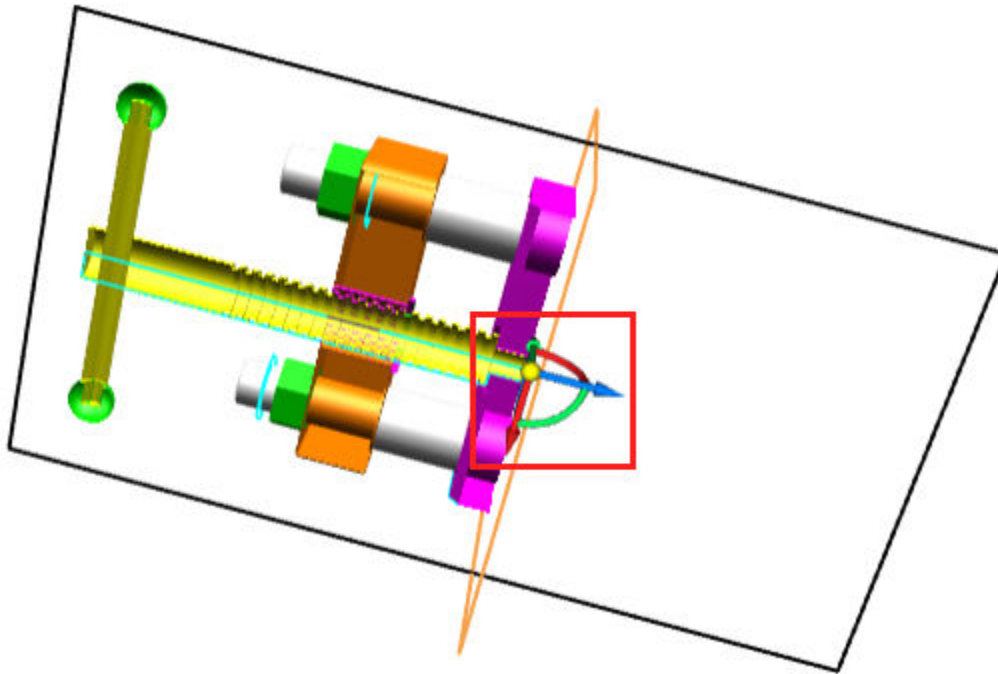
Place your cursor near the bounding edge of the selected section plane and press the right mouse button to access the section context menu.

To	Do this
Change the alignment of the section plane to match one of the three principal alignments (XY, XZ, or YZ).	<p>Choose the Orientation submenu.</p>

To	Do this
	
Control what portions of the assembly are cut away by the section plane.	Choose the Clipping Direction submenu.
Delete a section plane.	Select the section plane and do one of the following: <ul style="list-style-type: none"> • Click Delete . • Use the Delete key on your keyboard.
Move the section plane to a surface point on the assembly.	Choose Translate to Surface Point and select a point on the surface of the assembly. The section plane will update and move to that location without changing orientation.

Use the 3D manipulator

The 3D manipulator allows you to translate and rotate the selected section plane.




If no section plane is selected, the 3D manipulator is not displayed. If a section plane is selected, but the 3D manipulator is not visible, use the section plane context menu and choose **Show Manipulator**.

To	Do this
Translate the section plane.	<p>Click on one of the three axes of the 3D manipulator and drag the manipulator in that direction.</p> <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Note:</p> <p>The section name is changed to theta (Θ) in the Section panel.</p> </div> <p>As you drag the 3D manipulator along an axis, an entry field will appear. You can enter a precise value indicating how far you want to translate the 3D manipulator in that direction.</p> <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Note:</p> <p>Only the perpendicular axis will actually move the plane. The other two axes allow you to position the manipulator at different locations on the section plane. This is useful if you want to rotate the section plane about a point other than its center.</p> </div>
Rotate the section plane.	Select one of the three arcs on the 3D manipulator and drag it either along or parallel to that arc.



To	Do this
	<div style="border: 1px solid black; padding: 5px; margin-bottom: 10px;"> <p>Note:</p> <p>The section name is changed to theta (Θ) in the Section panel.</p> </div> <p>As you rotate the 3D manipulator, an entry field will appear. You can enter a precise value indicating how many degrees you want to rotate the 3D manipulator along the selected arc.</p> <div style="border: 1px solid black; padding: 5px;"> <p>Note:</p> <p>One of the arcs rotate in the plane of the section plane and will not change it's orientation. This is useful for adjusting the orientation of the 3D manipulator to change the alignment of the axes of the other two arcs so that you can rotate the plane about axes not aligned with the X, Y, or Z axis.</p> </div>

Delete 3D sections

1. From the work area toolbar, click **Section** .

The **Section** panel is displayed, listing all the sections under **Sections**.

2. Delete one or all sections.

To delete	Do this
A single section	<p>Select the section you wish to delete. Click  and confirm that you want to delete the section.</p> <p>Alternatively, with or without the Section panel displayed, right-click the section and select Delete from the context menu, or select the section and press Delete on your keyboard.</p>
All sections	<p>Click Delete All Sections  and confirm that you want to delete all sections.</p>

17. Monolithic files

About monolithic JT files and overridden subassemblies

A monolithic JT file is an assembly of individual components bundled together inside of a single JT file.

Your administrator must configure the Active Workspace environment to support monolithic JT files.

Prerequisite

Create a monolithic JT file and ensure the monolithic JT file name ends with **mono.jt**. For example, visemono.jt.

Viewing standalone monolithic JT files

Active Workspace treats standalone monolithic JT files which are not inserted into an assembly structure like part files. Any structure contained inside of the standalone monolithic JT file will not be represented in the structure tree, as the tree will not display when viewing standalone monolithic JT files. This is the same behavior seen when viewing any other standalone JT (part) file.

Viewing monolithic JT files in an assembly

The presence of an overriding monolithic JT file in an assembly causes the 3D viewer to replace any structure or geometry beneath that structure node with the contents of the monolithic JT file. This is advantageous because it allows the viewer to skip loading that data from Teamcenter, and instead load that data from the monolithic JT file, which is much faster. This override behavior only applies to the 3D viewer, and not to the structure tree. The structure tree still represents the content of the overridden subassembly as individual elements.

To see which component nodes are overridden by monolithic JT files, use the Arrange panel to add the **JT Override Children** column to the structure table.

Assembly lines whose substructure will be overridden by a monolithic JT file in the 3D viewer are indicated by **True** in the **JT Override Children** column. All other lines indicate **False**.

ID	JT Override Children	Revision Name
vise_moving_jaw_assm1	True	vise_moving_jaw_assm1
vise_guide_handle_assm1	True	vise_guide_handle_assm1
vise_jaw_plate1	False	vise_jaw_plate1
vise_plate_screw1	False	vise_plate_screw1
vise_plate_screw1	False	vise_plate_screw1


Overridden component visibility and selection behavior

Component visibility and structure element selection behave differently for subassemblies overridden by monolithic JT files. The structure tree still represents the structure that is overridden in the viewer. Selection of subassemblies or parts in the structure tree that are overridden by a monolithic JT file will cause all of the geometry contained in the monolithic JT file to appear selected (highlighted) in the 3D viewer. Clicking the visibility icon for any component within the overridden subassembly hides or shows all of the geometry contained in the monolithic JT file, and not just the geometry associated with the selected component.

Interacting with child nodes overridden by monolithic JT files

When interacting with a structure that contains elements or lines overridden by a monolithic JT file, the 3D viewer will only represent the content of the monolithic JT file. None of the individual JT files associated with lines in the structure that fall inside of the overridden subassembly will be displayed. This is true even if a line within the overridden subassembly has another monolithic JT file attached to it.

To interact with the geometry of overridden subassembly child nodes as individual components, open the subassembly in a new tab:

- Select the overridden subassembly within the structure tree and click **Open**  > **Open in New Tab**.

In the newly opened tab, the overriding monolithic JT file will be ignored and you can now interact with each child node of the subassembly individually. This will also allow you to use the structure tree to select individual child nodes and control their visibility in the 3D viewer.

Note:

Your administrator configures support for overriding JT subassembly nodes.

18. Working with sessions

About sessions

Sessions save the state of the viewer so you can later resume your work from where you stopped. Saving your work in a session enables you to preserve the following:

- Snapshots
- Quick Measurements
- 3D Measurements
- Sections
- Queries
- Part Orientations
- Workspace settings (floor options, material, etc.)
- Filtering and configuration applied to the product structure

Sessions are interoperable between Teamcenter lifecycle visualization and Active Workspace, which means, a session authored in one application can be opened, viewed, and modified in another.

When you open a session, you view the structure as it was previously saved retaining its applied filtering and configuration criteria. Depending on how your application administrator has configured the system, the session displays:

- Static data already stored in the session.

OR

- Dynamic data where the stored data is re-evaluated against the current data and the latest data is loaded.

You can override the value of this preference by creating a new instance of the preference at a higher-precedence location.

You can make **changes** to the session by applying new filters and configuration, and can also **save it as a new session**. Additionally, you can capture a snapshot of the product within the session to include the 3D measurement data applied on the structure. You can also **share a session** with other users.

Depending on how structures are indexed, you can perform the following tasks:




Scenarios		Tasks
Structures are indexed using Active Content Structure Indexing.	Structures are filtered.	Create a session. But the session will not retain the filters that are applied. Cannot visualize structures and capture snapshots.
	Structures are not filtered.	Create a session. Visualize large structures and massive structures, and capture snapshots.
Structures are indexed using Smart Discovery Indexing.	Structures are filtered.	Create a session. Visualize large structures and capture snapshots.
	Structures are not filtered.	Create a session. Visualize large structures and capture snapshots.

When a structure is indexed by using both types of indexing, select an appropriate revision rule to view the structure. To view the structure as indexed by Active Content Structure Indexing, select a revision rule above the rule separator. To view as indexed by Smart Discovery Indexing, select a revision rule below the rule separator.

Create a session

You can create a session for a regular structure and also for a structure within a workset with a partition scheme applied. You can save this session and share it with other applications.

To create a session:

1. Search for and **Open**  the desired structure.
2. Filter the structure as per your requirement.
3. Click **New**  > **Session** .
4. In the **Session** panel, specify the access level for other users:
 - a. Select the **Allow others to view** check box to provide *read* access to other users. Clear the check box to deny read access.
 - b. Select the **Allow others to edit** check box to provide *write* access to other users. Clear the check box to deny write access.

Note:

The **Allow others to edit** check box is displayed only when the **Allow others to view** check box is selected.

5. Click **Create** to create the session.

Search for a session

You can search for a session based on the properties associated with the top-structure node in that session. To quickly search for a session, you can further filter sessions by their properties such as **Name**, **ID**, and **Type**.

1. Search for sessions by performing a global search.

You can search with a keyword or the wildcards characters ***** or **?**.







The **Filters** panel is displayed.

2. In the **Filter** search box under the **Type** list, select the **Session** check box.

The session that you are searching for should be displayed in the panel to the right of the **Filters** panel.


You can further refine your search by applying more filters, such as **ID Contained in**, **Name Contained in**, and **Type Contained in** and many other filters.

Save a session as a new session

1. Search for a session.
2. Click **Open**  to open the session.
3. Click **Filter**  or select **Configure**  > **Configuration**  to modify the filters or the configurations applied on the session.
4. Click the session and then select **Session**  > **Save As** .
5. In the **Save As Session** panel, enter the required details and click **Save**.

View and update a session

1. Search for a session.




2. Click **Open**  to open the session. Your site administrator sets whether to load a session with the static data stored in the session or with the latest data.

Sometimes, when a session is opened, you get a message asking you to choose to restore the session. Due to some issues, such as getting logged out while making updates to a session, results in this message. Click **Restore** if you want to autorecover the updates that you earlier made to the session. If you do not want to restore, you can ignore the displayed message, and continue with your updates to the session.

Session information that gets restored includes snapshots, quick measurements, part orientation, 3D measurements, queries, sections, and workspace settings such as floor options and material.

3. To view the latest filters and configurations applied on the structure within the session, click **Replay**.

If you opened the structure in another window and edited it, the updates are not reflected in the session when you replay it. You must refresh the browser window to reload the changes in the session.



4. Click **Filter**  or select **Configure**  > **Configuration**  to modify the filters or the configurations applied on the session.

5. Click **Save Session** .

If the session is opened by another user at the same time and the other user made some changes to the session, you can choose to overwrite the session or save the session as a new one. You can overwrite the session only if your administrator has allowed overwriting of the session.

If another user removes some structure elements from the session, you receive a message with the list of the structure elements that are no longer available in the session.

6. To save a session as a new one:

- Click **Session**  > **Save As** .
- In the **Save As Session** panel, enter the required details and click **Save**.

7. To undo the updates to the session, click **Session**  > **Reset View** .

On resetting, the session goes back to its previously saved state.

Configure a session

Sessions are used to save the filtering and configuration criteria applied to a structure. When you open a session, you view the structure as it was previously saved retaining its applied filtering and configuration criteria.

When a new session is created for a structure, objects in the structure are displayed based on the configurations applied prior to creating the session. The status of the following **Show...** sets a preference. The value of the preference is not saved when the session is saved.

- Show Excluded by Effectivity
- Show Excluded by Variants
- Show Suppressed

You can use these **Show...** toggles in the **Configuration Settings** menu to control the display of the structure elements. You can apply any filters on the configured structures as required. The configuration is retained with the filters applied.

When the session is saved, the applied configuration (variant, effectivity, filtering, etc.) is also saved. However, the status of the **Show...** commands is not saved.


Show or hide occurrences excluded by effectivity

When a structure is loaded, all objects including those which are not effective for the specified revision rule are displayed by default. However, you can choose to display only those occurrences that are effective for a specified date, unit, range of dates, or unit numbers.

To show or hide the occurrences that are excluded by the currently applied effectivity, in the work area toolbar, click **Configuration Settings**  > **Show Excluded By Effectivity**.


Show or hide occurrences excluded by variant

When a structure is loaded, all occurrences including those occurrences which are not configured by the variant are displayed by default. However, a user can choose to hide or show the that are not configured.

To show or hide the occurrences that are excluded by the variant configuration, in the work area toolbar, click **Configuration Settings**  > **Show Excluded By Variants**.

Show suppressed occurrences



Occurrences in a structure can be hidden by setting the suppress property to *True*. When a structure is loaded, all occurrences including the suppressed occurrences are displayed by default.

To show or hide the suppressed occurrences, in the work area toolbar, click **Configuration Settings**  > **Show Suppressed**.

Share a session with other users

You can share a session with other users. As a prerequisite, the system administrator must set the **Has Class(FndOAppSession)** Access Manager rule. To share the session, you must be the creator of the session. After you share the session, other users can view or edit the session that is shared by you.

Make a session shareable while saving the session

1. Search for and open the structure.
2. Configure the structure as required.
3. Click **More Commands ...** > **New**  > **Create Session** .
4. In the **Create Session** pane, specify the access level for other users:
 - a. Select the **Allow others to view** check box to grant *read* access to other users. Clear the check box to deny read access.
 - b. Select the **Allow others to edit** check box to grant *write* access to other users. Clear the check box to deny write access.



Note:

The **Allow others to edit** check box is displayed only when **Allow others to view** check box is selected.

5. To create the session, click **Create**.

Initiate a workflow to share a session

When you share a session, you must inform the person you have shared the session with. You can communicate using any method such as emailing the session URL. You can also use a workflow to share sessions and assign tasks to other users.

1. Search for and select the session that you want to share.
2. Click **More Commands ...** > **Manage**  > **Submit to Workflow** .

The **Submit to Workflow** panel with a list of workflow templates is displayed.

3. Select the **Session Collaboration Workflow** template.
4. Enter a description for the workflow participants, and select the appropriate workflow template.

A default workflow is defined for sharing the session. It is automatically selected as the workflow template.

5. Click **Assignments** to assign users and resource pools to the workflow.





Note:

You must have selected the **Allow Others to View** check box while creating a session in order for the assigned user to receive an email about the shared session.

6. Click **Submit**.

Release a session

To set a *release* maturity on a session:


1. Search for the session you want to release.
2. Click **Open**  to open the session.
3. Click **More Commands**  > **Manage**  > **Submit to Workflow** .

The **Submit to Workflow** panel with a list of workflow templates is displayed.

4. Select the **Session Release Workflow** template.
5. Enter the required details and click **Submit**.

Find an element within a session

You may want to search for a structure element within a session. Searching for an element is not limited to the filtered structure displayed in the session. The element is searched for across the unfiltered structure.

1. Open a session.
2. Click **Find** .
3. Enter the search keyword.
4. (Optional) To narrow your search, select **Find within** to find the element within a certain assembly.

Capture a snapshot of a session

You can capture a snapshot of a session.



Snapshots capture the 3D data associated with your product. When you load the product in the 3D viewer and take a snapshot, you capture the current 3D view, including camera, visibility, selection, view port (pan, zoom, and rotate modes), orientation, sections, measurements, queries, and markups. You also capture the configuration and filtering criteria.

You can capture a snapshot of the current view of an assembly as an in-session snapshot. In-session snapshots are contained within the session. You cannot share these snapshots independently with other users.

View a session in other applications


To easily retrieve a structure that you are currently working with, you save it within a *session*. The session also retains any filters and configurations applied to the structure. You can view a saved session and update the structure in applications such as Teamcenter lifecycle visualization, NX, or CATIA.

When you make changes to a session, and then open the unsaved session in other applications such as Lifecycle Visualization, NX, or CATIA, the changes must be saved or discarded before opening the structure in the other application. You can save or discard the unsaved changes.

1. Search for a session that you want to view.
2. Click **Open**  to open the session.
3. To work with the 3D data associated with the structure, open the session in Teamcenter lifecycle visualization by clicking **Open**  > **Open in Visualization**.

Before opening the session in Teamcenter lifecycle visualization, you must first replay and save the session in Active Workspace.

Caution:


- Active Workspace viewer and Teamcenter lifecycle visualization have different default rendering settings. Due to these differences, you may see a difference in lighting while working with an interoperable session.
- A multi-document AppSession created in Teamcenter lifecycle visualization is not supported in Active Workspace. Multi-document AppSession files can be identified in Active Workspace by the following icon .

Note:

Your administrator must define the necessary configuration settings if you are using:

- Teamcenter lifecycle visualization 12.4 or earlier along with a Teamcenter version later than 12.4.
- Teamcenter lifecycle visualization 12.4 or earlier along with Active Workspace 5.0 or later.

Teamcenter lifecycle visualization versions 12.4 and earlier do not understand session information saved by Active Workspace, but can still load the configured assembly referenced by the session.

4. To make design modifications to the structure, open the session in CATIA or NX by clicking **Open**  > **Open in CATIA** or **Open in NX**.

19. Capturing and managing 3D data using snapshots

About Visualization snapshots

Snapshots capture the 3D data associated with a product. When you load a product in the 3D viewer and take a snapshot, you capture the current 3D view, including camera, visibility, selection, view port (pan, zoom, and rotate modes), orientation, sections, measurements, queries, and markups. You also capture the configuration and filtering criteria.

You can capture snapshots of several different views of the product and can return to any of these by applying its corresponding snapshot. When **applied**, the snapshots restore the configuration and the view state as captured in it. Additionally, snapshots enable you to share 3D data and collaborate with other users and business stakeholders.

You can capture the 3D data associated with a product in the following ways:

- **Capture snapshots of a product**

When working with products, you can **capture snapshots** to preserve the BOM configuration and view state and save it for later use. Additionally, you can share these snapshots individually with other users using *Discussions*.

- **Capture snapshots of a product within a session**

When working with sessions, you can **capture snapshots of an assembly** to preserve the state of the current view within a session and **save them** for later use. Snapshots are contained within the session and you cannot share these independently with other users.

- **Capture snapshots of a workset**

When working with **worksets**, you can capture snapshots of a workset to preserve the working context and save it for later use. Additionally, you can share these snapshots individually with other users using *Discussions*.

Note:

If a subset within a workset has any filters set on it, these filters are not captured in a workset snapshot.

You can perform the following operations on snapshots of products and worksets:

- Capture snapshots of several different views of the product (or assembly) or workset and return to any of these views by **applying** its corresponding snapshot.

- **Modify** an existing snapshot by renaming or updating it, or deleting it when no longer required.
- **Share** a snapshot with other users.

Working with product snapshots

Capture a snapshot


In Active Workspace, you can capture the following snapshots:

- **Product snapshots**

To preserve the BOM configuration and view state and save it for later use.

- **Workset snapshots**

To preserve the working context and save it for later use.

1. **Open the product or assembly** or workset in the 3D viewer in Active Workspace.
2. From the work area toolbar, click **Capture Snapshot** .

The snapshot is created and a success message is displayed.

Note:

If a subset within a workset has any filters set on it, these filters are not captured in a workset snapshot.

3. Click **Gallery** .

The snapshot is displayed under the **Snapshots** tab of the **Gallery** panel.

By default, the snapshot is assigned a system-generated title. Based on your requirement, you can **rename** the snapshot.

Note:

You can also find the snapshots by clicking the drop-down list in **Snapshot source** and selecting from the category of snapshots.

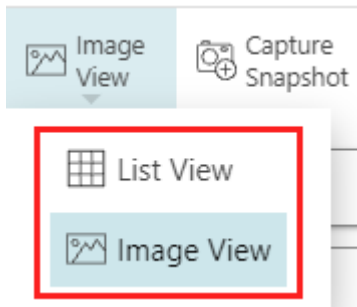
- **Created By Me**

- **Shared By Me**
- **Shared With Me**

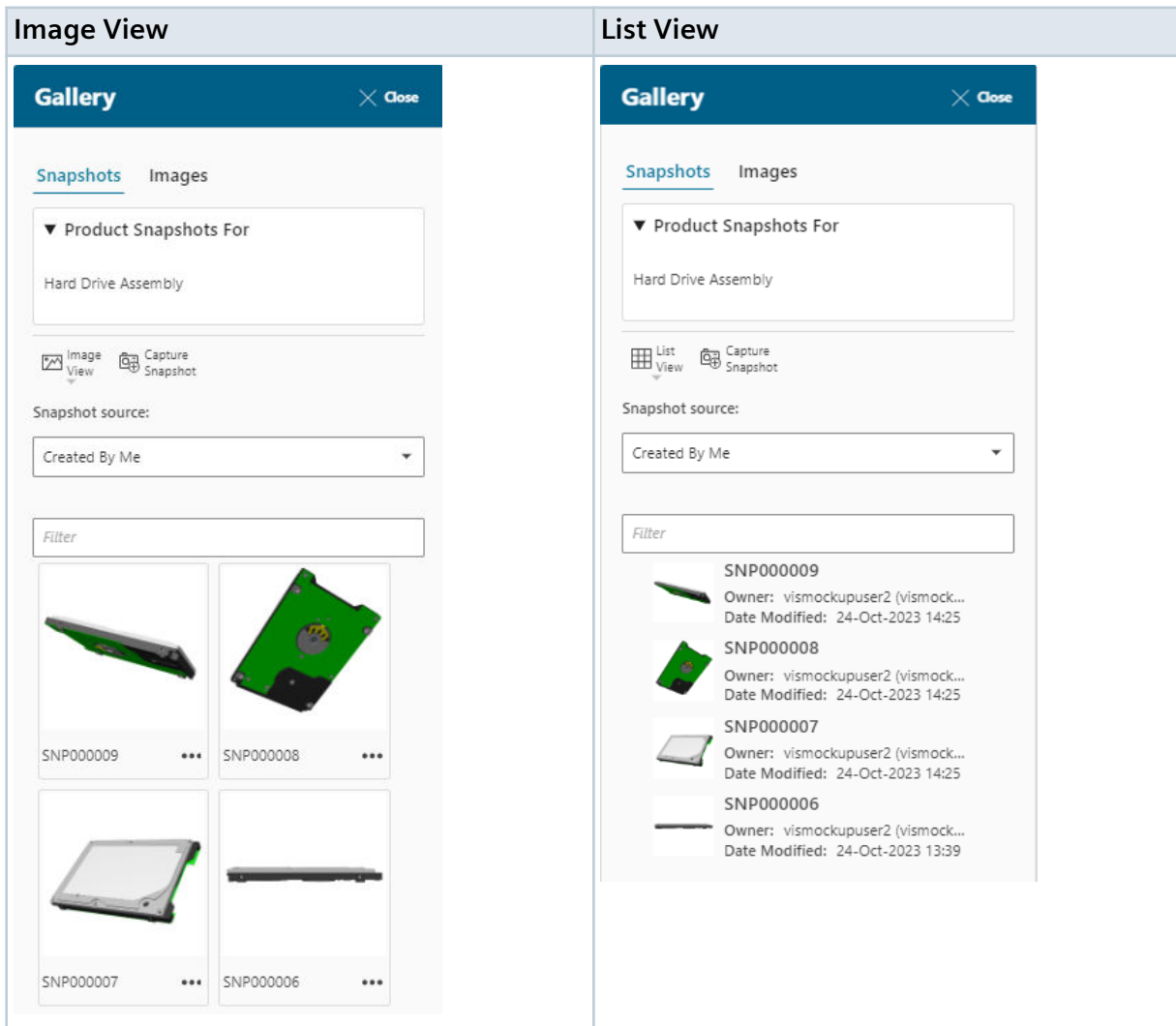
4. To take another snapshot from the **Gallery** panel, click **Capture Snapshot** .

The new snapshot appears under the **Snapshots** tab.

5. By default, the snapshots are displayed in the gallery in **Image View**.





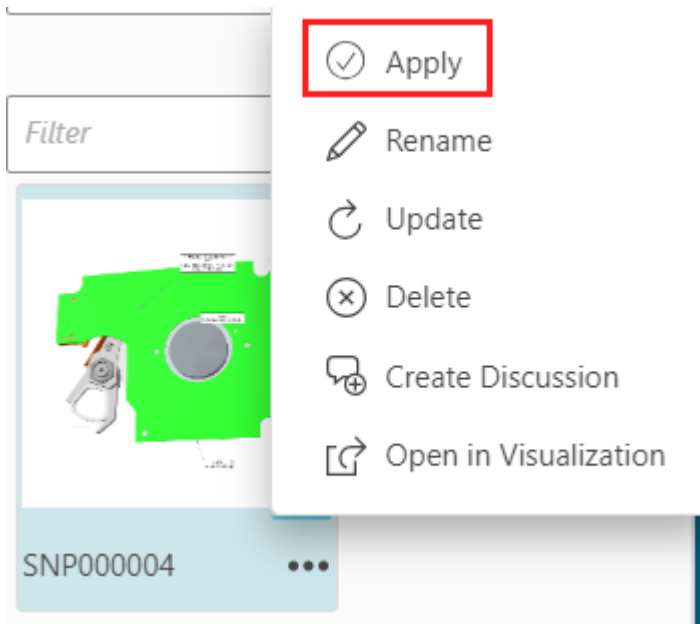
To view the snapshots as a list, click **Image View** and from the list, select **List View**.



Restore the state of an assembly or workset from a snapshot


You can restore a specific captured BOM and 3D state of an assembly or the working context for a workset from a snapshot to the assembly or workset in the 3D view.

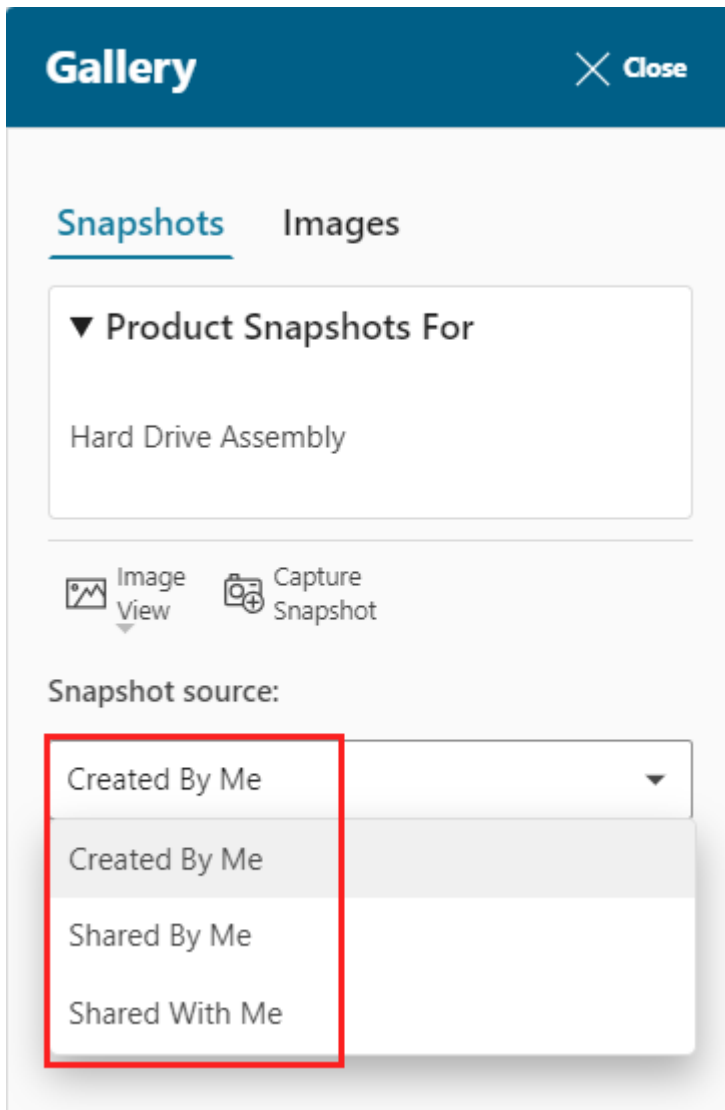
1. **Open the product or assembly** or workset in the 3D viewer in Active Workspace.
2. From the work area toolbar, click **Gallery** .
3. In the **Snapshots** tab of the **Gallery** panel, select the category of snapshots that contains the snapshot whose BOM and 3D state or working context you wish to apply to the assembly or workset.
4. From the snapshot thumbnails that are displayed, select the snapshot whose BOM and 3D state or working context you wish to apply to the assembly or workset and click **More Commands** **...** > **Apply** .



The BOM and 3D state of the selected snapshot is applied to the assembly in the 3D viewer. In case of worksets, a working context is applied to the workset in the 3D viewer.




Modify a snapshot

1. **Open the product or assembly** and click the 3D view.
2. From the work area toolbar, click **Gallery** .
3. In the **Snapshots** tab of the **Gallery** panel, from the drop-down list in **Snapshot source**, select the category of snapshots that contains the snapshot which you wish to modify.



- From the snapshot thumbnails that are displayed, select the snapshot that you wish to modify and click **More Commands** **...**.

Select the action that you wish to perform.


Command	Purpose
Rename  Alternatively, double-click the snapshot name box	By default, the snapshot is assigned a system-generated title. You can rename the snapshot based on the naming convention followed by your organization.
Update 	Replace the selected snapshot state to reflect the current BOM and 3D state of the assembly in the 3D view.
Delete 	Remove the snapshot from the list of snapshots for the assembly.

Command	Purpose
	<div style="border: 1px solid black; padding: 5px;"> <p>Note:</p> <p>You can only delete those snapshots that are created by you and are not currently a part of any discussion.</p> </div>

Share a product snapshot with other users

You can use the *Discussion* feature in Active Workspace to share snapshots created by you with other users to enable them to provide their inputs. Additionally, you can use *Discussions* to provide your inputs for snapshots shared with you by other users.


You can start a discussion for a snapshot using any of the following options:

- From **MY GALLERY** tile > **My Gallery** page
- From the 3D viewer > **Gallery** on the work area toolbar
- From the **Discuss**  command on the primary toolbar

To start a discussion

1. **Open an assembly (product)** in the 3D viewer.

Alternatively, on the home page, click the **MY GALLERY** tile.

2. In the **3D** viewer, on the work area toolbar, click **Gallery** .
3. In the **Snapshots** tab of the **Gallery** panel, from the drop-down list in **Snapshot source**, select the category of snapshots that you wish to discuss with other users.

- **Created By Me**
- **Shared By Me**
- **Shared With Me**

Alternatively, if you chose to open **My Gallery**, select the tab from which you wish to choose the snapshot to share.

4. From the snapshot thumbnails that are displayed, select the snapshot which you wish to share and click **More Commands** **...**.

5. Select **Create Discussion**.

The **Discuss** panel is displayed.

Note:

If there are existing discussions on the snapshot, you can choose to open and edit the discussion for that snapshot.

6. Click **Add Participant** ⊕ to add a participant to the discussion, for example, Manny. Select the participant and click **Add**.
7. (Optional) Click the **Add Source** ⊕ to add an additional source object to the discussion.
8. (Optional) Click the **Private Message** box to restrict the participants in the discussion. Only the selected participants can participate in the discussion. In addition, participants in a private message automatically follow the discussion. When created, private messages appear with a lock icon.
9. (Optional) Click the **Tracked** box to track the status and priority of a discussion. By default, the status is **Open** and the priority is **Low**. You can set the priority (**Low**, **Medium**, or **High**) using the dropdown list.

Note:


You can track a discussion only if your administrator has configured this feature. Otherwise, the **Tracked** check box is not displayed.

10. (Optional) Click **Remove Snapshot** ⊖ to remove the snapshot from the discussion.

Note:

Once you remove an existing snapshot, to include a snapshot in this discussion, click **Capture Snapshot** in the **Snapshot** area. This creates a 3D snapshot in the **Discuss** panel. The snapshot is also added to the **Gallery**. You can also access these from the **MY GALLERY** tile on your home page.

11. Type your message in the **Message** field.
12. Click **Create** to create the discussion on the object.
13. The new discussion appears on the **Discuss** panel. It also appears in the **Tracked** tabs in **Discussions**. Each discussion has a timestamp and new discussions appear at the top of the list. As the creator of the discussion, you are automatically set to follow all replies in the new discussion.

To review the discussion for a particular snapshot, choose the snapshot, and select **More Commands ...** > **Open Discussion** .

View snapshots

You can use the following options to view snapshots:

- **My Gallery** on the home page in Active Workspace: Displays all snapshots of all assemblies and worksets.

Note:

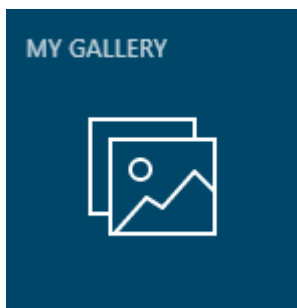
The **My Gallery** tile will be available on your home page in Active Workspace only if:

- Your administrator has installed the Active Workspace **3D Visualization** module.
- Your administrator has configured the **My Gallery** tile for your workspace. By default, the **My Gallery** tile is available for the Author, Consumer, and the Default workspaces.

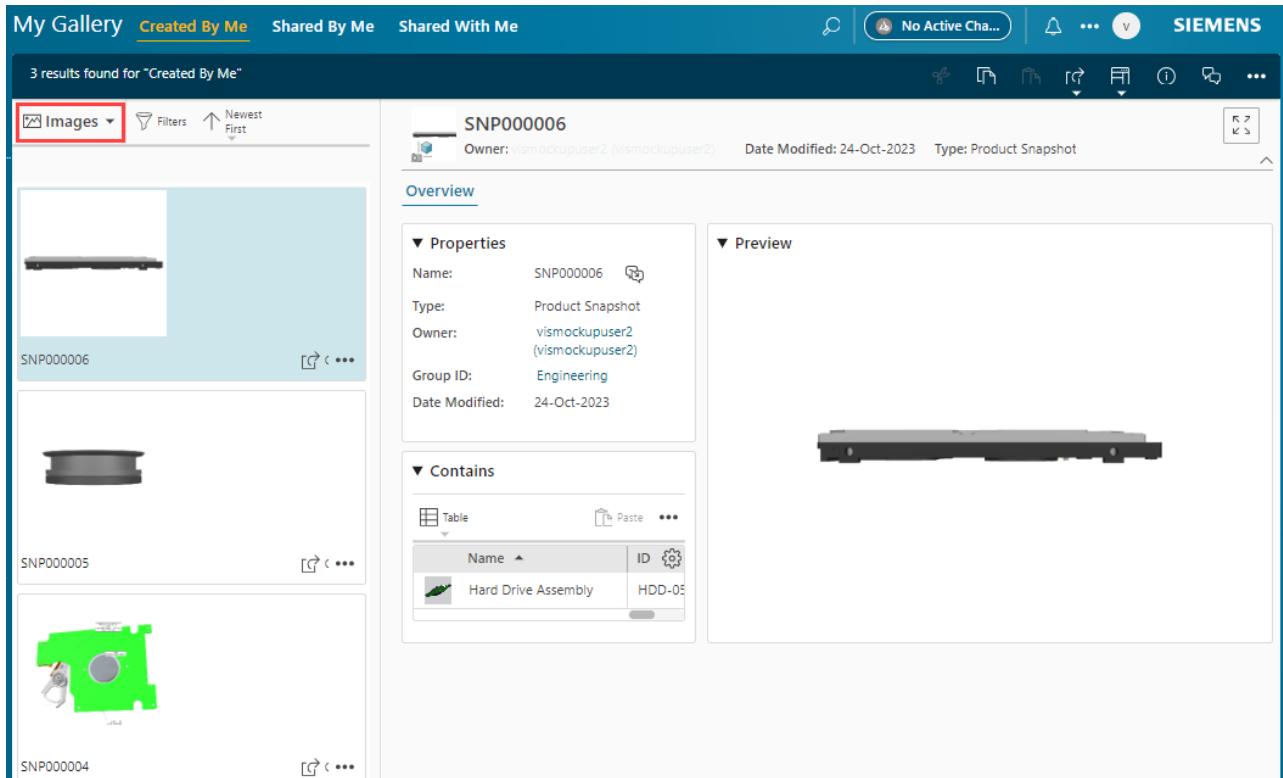
- **Gallery** on the work area toolbar in the **3D** view of the assembly: Displays snapshots for the assembly or workset open in the 3D viewer.

View snapshots using My Gallery

1. On the home page, click the **MY GALLERY** tile.




A list, containing all the snapshots is displayed. By default, the snapshots are displayed as large images arranged in a grid.

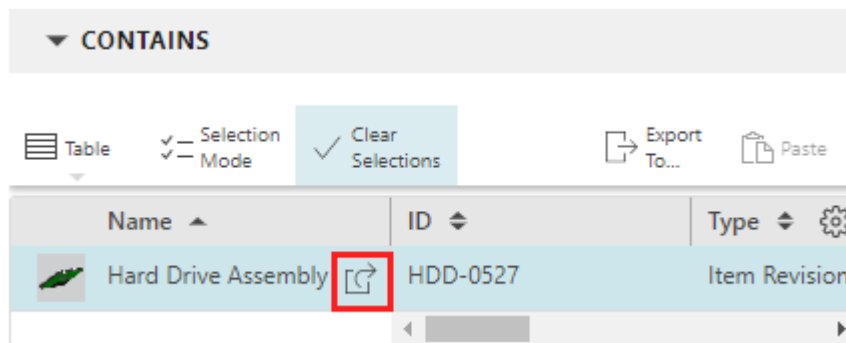



You can choose whether the list of snapshots is displayed: as **Images** or in a **Table**.

To view	Perform this action
All product snapshots that are created by you	Click Created By Me .
All product snapshots that are shared by you with other users using discussions	Click Shared By Me .
All product snapshots that are shared with you by other users using discussions	Click Shared With Me .

- To view the details about a particular snapshot, select it from the snapshot list. The details are displayed in the **Overview** tab.

You can open the product associated with the selected product snapshot by selecting it in the **Contains** section and clicking **Open** .



3. To narrow your snapshot list, click **Filters**  and choose from the available filters.
4. To view a sorted list of snapshots, click any one of the options. The snapshots are sorted:
 - From new to old (or vice versa) based on the time and date of their creation. By default, the list is sorted in the order of **Newest First**.

To change the order, click **Newest First** and from the dropdown, select **Oldest First**.


 - From **Ascending** to **Descending** based on the name of the snapshot.
5. To view the product or workset contained in the snapshot in a new window or a new tab in your browser or in Lifecycle Visualization, select the snapshot and click **More Commands ...**

Note:

The *Open* commands open the product contained in the snapshot and applies the state (BOM configuration and view state) as captured in the snapshot to the product. In case of snapshots of worksets, if you choose to view the snapshot in Lifecycle Visualization, **Open in Visualization** is enabled only for the entire workset and not for the subsets within the workset.

View snapshots from the 3D Gallery

In the **3D** view, you can access snapshots of the product (assembly) or workset that is currently open.

1. From the work area toolbar, click **Gallery** .

A list, containing all the snapshots that you created is displayed. By default, the snapshots are displayed as large images arranged in a grid.

Note:

The assembly for which the list of snapshots is displayed is specified under **Product Snapshots For**.

You can choose how the list of snapshots is displayed: as **Images** or in a **Table**.

To view	Perform this action
All product snapshots that are created by you	Click Created By Me .
All product snapshots that are shared by you with other users using discussions	Click Shared By Me .
All product snapshots that are shared with you by other users using discussions	Click Shared With Me .

- To narrow your product snapshot list, specify the **Filter** criteria.
- To view the list of actions available for a snapshot, select the snapshot and click **More Commands**
...


Working with snapshots in a session

Capture an in-session snapshot

You can capture a snapshot of the current view of an assembly as an in-session snapshot. In-session snapshots are contained within the session and you cannot share these independently with other users.

- Load the assembly** in the 3D viewer.
- Create a new session**.

The session is created and loaded in the 3D viewer.

- Perform the actions that you want to capture in a session, such as adding markups, sections, or measurements.
- From the work area toolbar, click **Capture Snapshot** .


The snapshot is created and a success message is displayed.

- Click **Gallery** .

The snapshot is displayed under the **Snapshots** tab of the **Gallery** panel.

Note:

The session for which the list of snapshots is displayed is specified under **Session Snapshots For**.



By default, the snapshot is assigned a system-generated title. To rename the snapshot, select it, click **More Commands** **...**, and click **Rename** . Alternatively, you can double-click the snapshot name field and rename the snapshot.

Enter the new name and click **Update**.

- To take another snapshot from the **Gallery** panel, click **Capture Snapshot** .

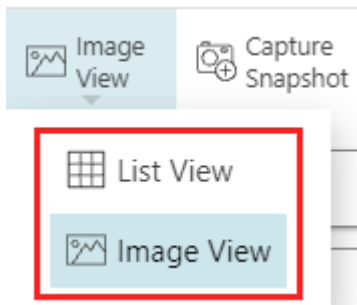
The new snapshot appears under the **Snapshots** tab.

Note:

You can load different views of a product, add markups, sections, measurements, and capture multiple snapshots throughout an active session. All snapshots currently loaded in the viewer can be viewed in the **Gallery** . To go back to a particular view, apply the respective snapshot from the **Snapshots** tab in the **Gallery**  by selecting it.

- By default, the snapshots are displayed in the gallery in **Image View**.

To view the snapshots as a list, click **Image View** and from the list, select **List View**.

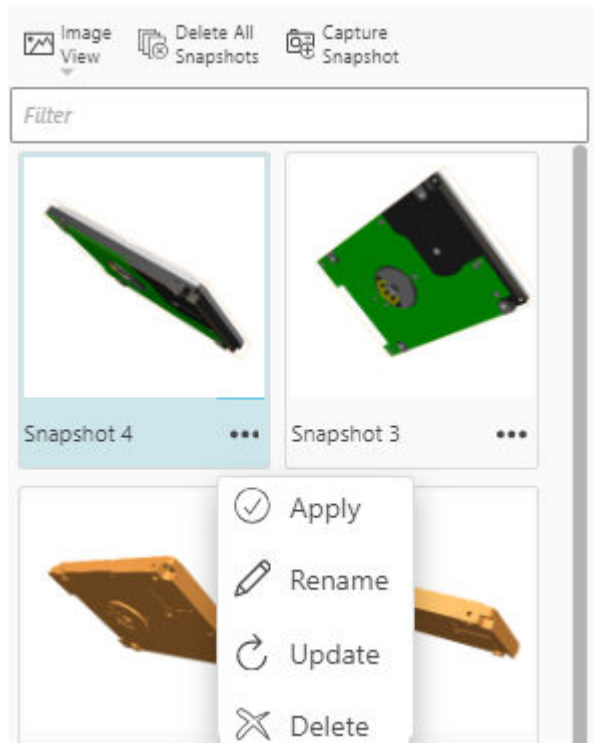


- To view a list of all actions that you can perform on the snapshot, select the snapshot and click **More Commands** **...**:

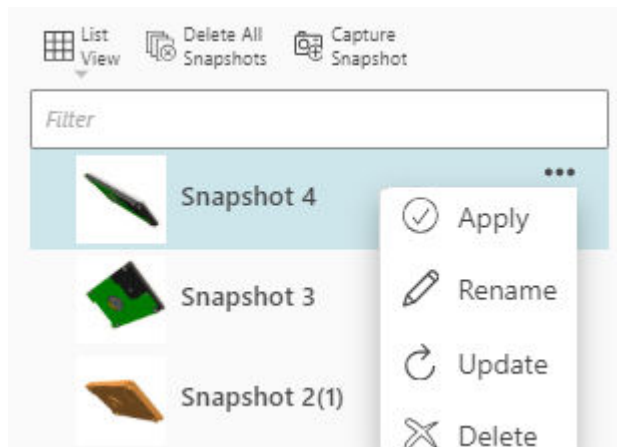
Note:

The **Apply** command applies the BOM and 3D state of the selected snapshot to the in-session assembly in the 3D view.

- In the **Image View**




- In the **List View**




Save snapshots in a session



You can save **captured snapshots** within new or existing sessions.

To save snapshots within an existing session

1. Search for a session using the global search or **Quick Access**.
2. Click **Open**  to open the session.

3. **Capture snapshots.**
4. Click **Save Session** .

Tip:

To save the session as a new one, click **Session**  > **Save As** .

In the **Save As Session** panel, enter the required details and click **Save**.




The snapshot is saved within the new session you created from the existing session.

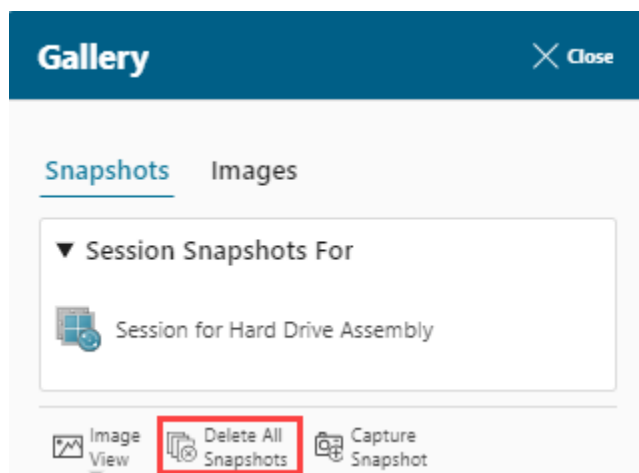
5. Snapshots captured within a session can be viewed when the **session is opened in other applications** such as Teamcenter lifecycle visualization and NX.

Note:

Session information gets autosaved at appropriate times during a working session. If you capture a snapshot within a session and then log off or the session results in a timeout, when you reload the structure or the saved session, the snapshot is automatically restored.

Delete a snapshot

- To delete a single snapshot, select it in the **Gallery** panel, click **More Commands** , and then click **Delete** .
- To delete all snapshots related to the currently loaded product, click **Delete All Snapshots** . This deletes all snapshots of the product captured in the current session.





Working with visual reports

You can view visual reports for an assembly in the 3D viewer to visually grasp complex information about many parts quickly.

Procedure

1. Open an assembly from Teamcenter in standalone Lifecycle Visualization.
2. Create and apply a visual report in standalone Lifecycle Visualization.

For more information about Visual Reports, see *Working with visual reports* under Teamcenter Lifecycle Visualization.

3. Capture a 3D Snapshot of the applied visual report and save it to Teamcenter as a Product Snapshot, or save a Collaborative Session containing that 3D Snapshot to Teamcenter.
4. In Active Workspace, find and open the Product Snapshot or the Collaborative Session containing the 3D Snapshot of the Visual Report.
5. From the work area toolbar of the 3D viewer, click **Gallery** .
6. In the **Snapshots** tab of the **Gallery** panel, find the 3D Snapshot containing the Visual Report information created using standalone Lifecycle Visualization.
7. Click **More Commands** **...** on the 3D Snapshot containing the visual report results and select **Apply** .

The visual report results captured by the 3D Snapshot are applied and displayed in the 3D viewer.

20. Visualizing structures in partitions

Access data within partitions

A BOM architect organizes a large product into smaller, accessible sections called *partitions* and arranges logically and hierarchically within the partitions. Partitions can belong to different partition schemes such as functional, physical, or spatial.

For information on viewing and managing structures, see *Structure Management on Active Workspace — Usage*.

For information on partition schemes and partitions, see *Structure Partitions — Usage*.

Visualize structure elements located within partitions

To visualize the structure elements located within a partition:

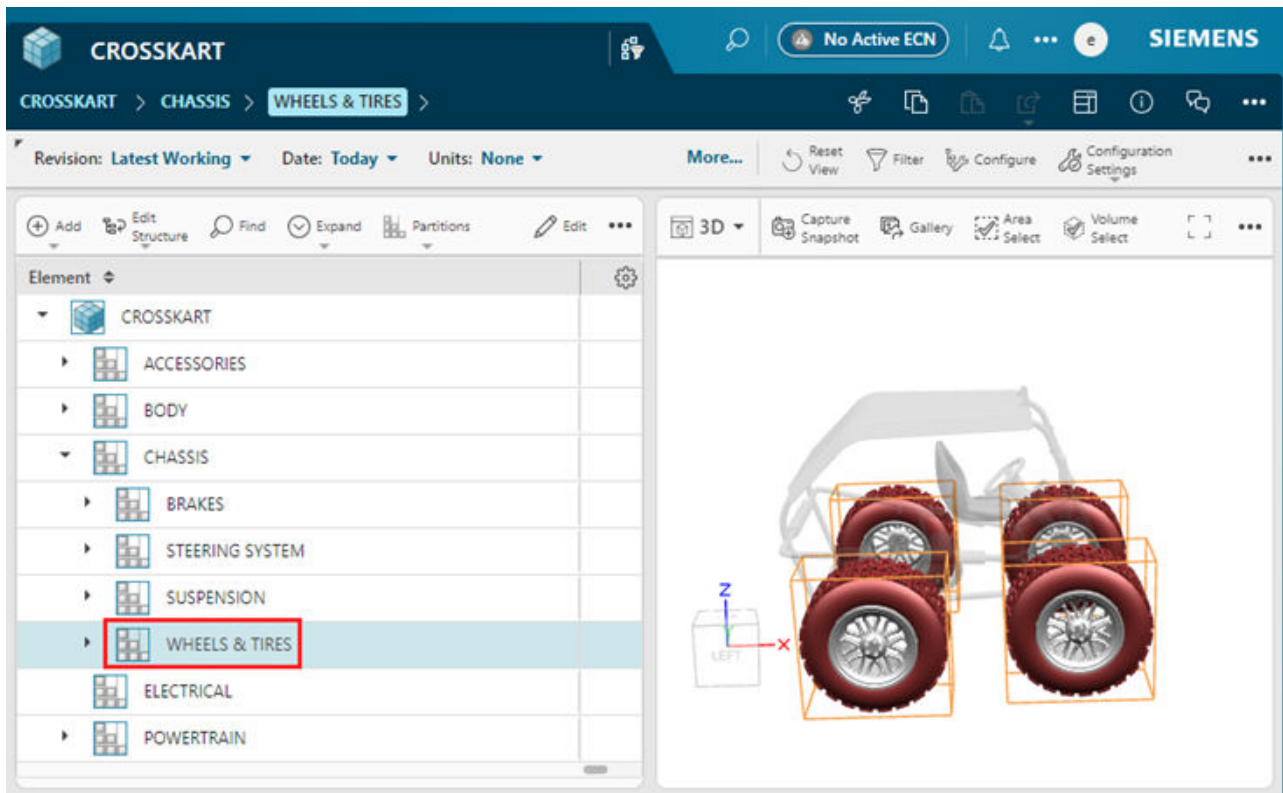
1. Open the structure that you want to visualize. By default, the structure opens in the partition scheme that is set as the primary scheme.

Note:

Ensure that a partition scheme exists before you open the structure that you want to visualize. For information on partition schemes and partitions, see *Structure Partitions — Usage*.

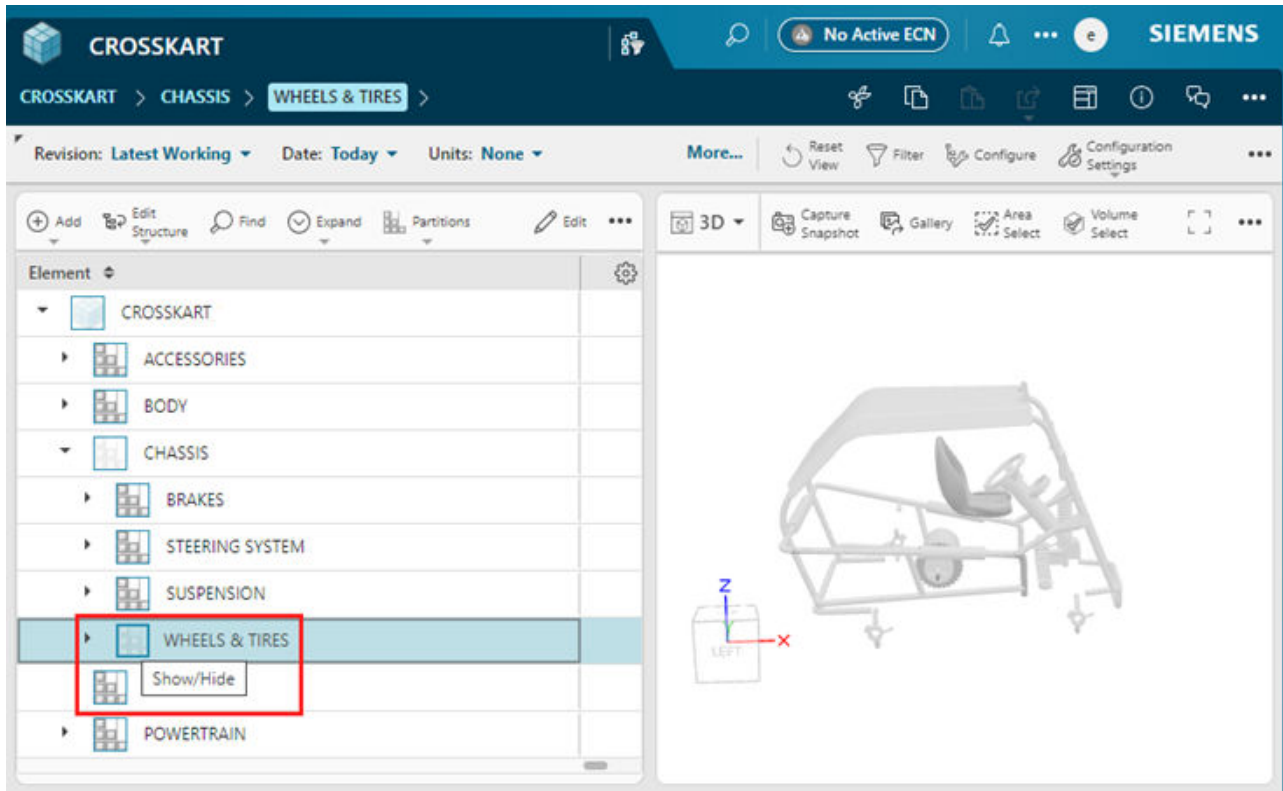
2. Select the required scheme from **Partition Scheme** that contains the required partitions.
3. To visualize the structure elements located within a partition, select the partition, and go to the 3D viewer.

All the elements within the selected partition are highlighted with their bounding boxes.



If the selected partition has in turn a child partition within it, all the structure elements within the child partition are also highlighted in the 3D viewer.

You can choose to not visualize the elements of a specific partition by clicking the **Show/Hide** icon. For example, the icon is displayed next to WHEELS & TIRES, as shown in the following image. To visualize the elements again, click the **Show/Hide** icon.



When you set variant criteria on partitions, the structure is configured to either show or hide the partitions and the elements within. Accordingly, the elements are displayed in the 3D viewer as well.

In several cases, the configured structure and the 3D viewer might not be synchronized.

- **Scenario 1**

For a structure in a *workset*, a *Functional* scheme is applied and contains a variant criterion, VC1. In this case, the structure elements with a different variant criterion, VC2, are not included in the structure. However, the 3D viewer shows these elements. When you switch to a different partition scheme, the same scenario applies. In such cases, for structures in a workset, you can synchronize the partitioned structure and the 3D viewer by clicking **Replay**.

- **Scenario 2**

A component resides in two different partitions. It is included in the structure configured for one partition, whereas it is not included in the structure configured for the other. When such a mismatch occurs, the component is displayed both in the partitioned structure and in the 3D viewer.

- **Scenario 3**

In a partition scheme, structure elements at different hierarchical levels reside in two different partitions. When you set variant criteria on a partition to configure the structure such that the partition is not included in one case, the structure and the 3D viewer show a mismatch. Siemens

recommends configuring the system to restrict the addition of structure elements in partitions in such a manner.

Filter structures by partitions

You filter a product structure to work with a specific product definition, which is comparatively small. However, for this to work, your system administrator must index the structures using Smart Discovery Indexing.

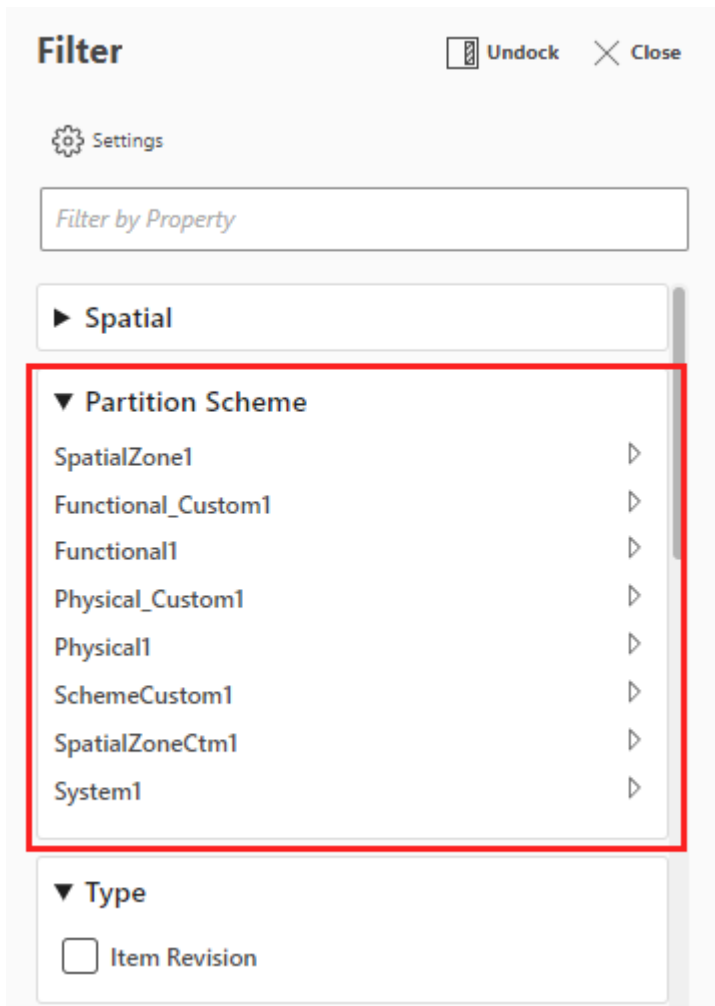
To filter a structure by partitions:

1. Open the structure that you want to filter.

If a primary partition scheme is set by the administrator, the structure opens in this scheme, by default. However, if you have overridden the preference set by the administrator, the structure opens in the partition scheme that you have set.

2. Click **Filter** .

The **Partition Scheme** section on the **Filter** panel lists the partition schemes created for this structure.



3. Select a partition scheme from the list.

The **Filter** panel displays the partition hierarchy within the selected partition scheme.

Filter Undock Close

Settings

Filter by Property

► Spatial

▼ Partition Scheme

- SpatialZone1
- Functional_Custom1
- Functional1
- Physical_Custom1
- Physical1
- SchemeCustom1
- SpatialZoneCtm1
- System1

▼ Type

Item Revision

► Owner

► Group ID

► UPC

► VPPS

► Family Address

Partition Scheme: Functional1

Filter by Name

Name	ID	D
<input type="checkbox"/> FunPtn1	PTN015075	
<input type="checkbox"/> FunPtn2	PTN015076	

Back Apply

To select the required partition, you can either navigate the partition hierarchy by expanding the tree or by searching for a partition. You can expand a partition and choose a child partition. You can also search for a specific partition in the **Filter by Name** box and select the partition from the results list.

To go back to the partition hierarchy (tree), in the **Filter by Name** box, click ✕ and then click 🔍. When you go back to the tree, by default, you do not see the partition as expanded, unless you have already expanded it. That is, when you select a partition, the partition hierarchy does not expand automatically up to that level and show you the selection. You have to manually expand the parent partition and then see the partition that you selected.

Teamcenter saves the partition expansion state in a partition scheme. When you reopen the partition scheme, you can see the previous expansion state.

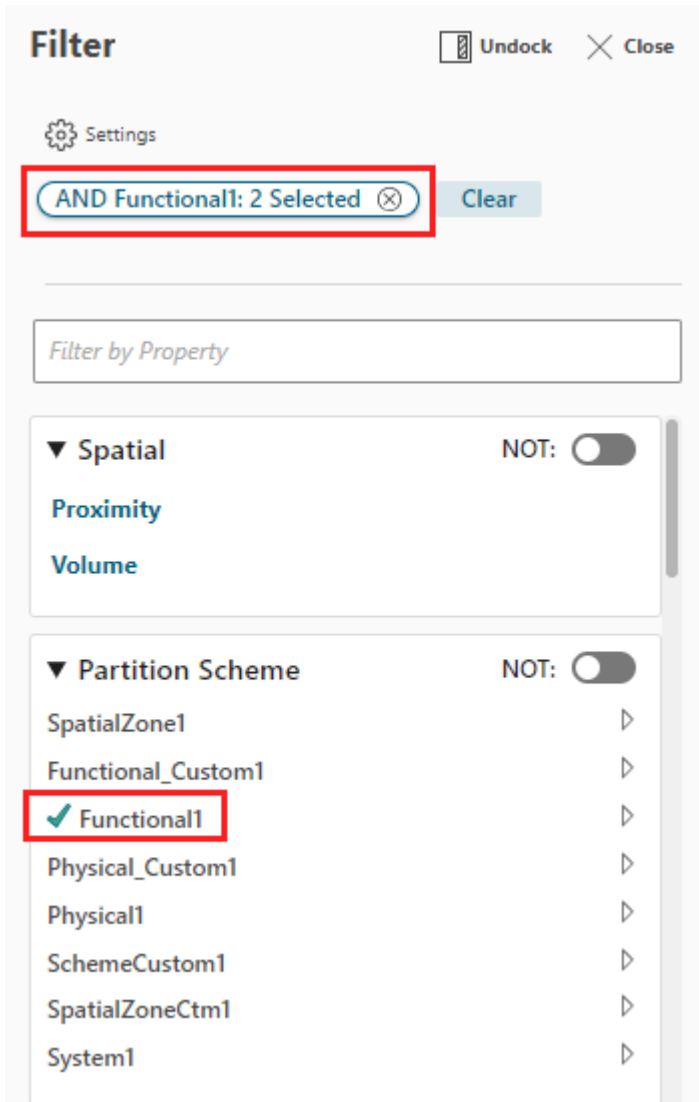
4. Select the required partitions and then click **Apply**.

Partition Scheme: Functional1

Filter by Name

Name	ID	Description	Type
<input type="checkbox"/> FunPtn1	PTN015075		Functional Part...
<input type="checkbox"/> FunPtn3	PTN015079		Functional Part...
<input checked="" type="checkbox"/> FunPtn4	PTN015081		Functional Part...
<input type="checkbox"/> FunPtn2	PTN015076		Functional Part...
<input checked="" type="checkbox"/> FunPtn5	PTN015083		Functional Part...
<input type="checkbox"/> FunPtn6	PTN015084		Functional Part...

A filter expression is generated and is displayed at the top of the **Filter** panel. Also, in the **Partition Scheme** section, a tick mark is displayed for the partition schemes from which the partitions are selected for filtering.



To exclude partition schemes as a filter, turn on the **NOT** toggle.

The loaded structure is filtered in real time based on the applied filters.

If you do not want the structure to be filtered in real time while you are applying filters, you can turn off **Auto-apply filters** from **Settings** ⚙️. To load the filtered structure in this case, click the **Filter** button on the **Filter** panel.

If a selected partition, in turn, has child partitions, the elements of the child partitions are also considered in the filter criteria.

5. You can also filter the structure by using a combination of volume, proximity, and attributes if you have the Smart Discovery for Structures license.
6. To remove a filter, click ⊗ for the partition scheme.

To remove all filters, click **Clear**.

Tip:

You can save the filtered structure in a session, workset, or a product snapshot for easy retrieval. When you open these later, they retain the filters and scheme applied to the structure.


21. Working with product views

Search for an issue report revision


In Active Workspace, you can search for an issue report revision created in Teamcenter and view the visualization data associated with it.

Tip:

All issues assigned to you for review will be available in your **INBOX**.

To view an issue assigned to you, on the Active Workspace Home page, click **INBOX**, select the issue, and click **Open** .

To find an issue report revision in Active Workspace:

1. On the Home page, in the **Search** bar (global search), enter the name of the issue you are looking for and click .
2. Alternatively, you may search for the desired issue by performing an advanced search:
 - a. Click the **Advanced Search** link below global search to display the **Advanced Search** panel.
 - b. From the *Item ID* drop-down list, select **General**.
 - c. Click **Clear All**.
 - d. In the **Type** list, enter the keywords **Issue Report Revision** and select the **Issue Report Revision** option.

Advanced Search Undock Close

All Public Private

General...

Preferred:

Clear All

Name:

Description:

Type:


Issue Report Revision

Owning User:

Owning Group:

Search

e. Click **Search**.

3. Select the issue from the search results and click **Open** .

Depending on your requirement, you may choose from any of the methods of searching: *Global search* or *Advanced search*.

Preview product views

The product view associated with an issue or issue report revision is available in the **Attachments** tab. There can be more than one product view attached to an issue report revision.

To preview the product views associated with the issue:



1. **Open the visual issue.**

2. Click **Attachments**.

The product views associated with the issue are displayed. They usually include markups and give a preview of the problem that needs to be fixed.

3. To view the snapshot view data associated with the issue, select the product view (**Snapshot Before Fix** or **Snapshot After Fix**).

Based on your selection, a high-quality preview of the selected object is displayed in the **PREVIEW** section.

4. To directly open the snapshot view data in 3D, select it from the product view list, click **Open**  on the Primary toolbar. From the drop-down list, select **Open in New Tab** .





Note:

The **Open in New Tab** and **Open in New Window** options are not available when using Active Workspace Application Connect or in the hosted mode.

The snapshot view data opens in the 3D viewer. Here, you can create measurements, sections, change viewer options, manage visibility controls. However, you cannot capture 2D images or 3D snapshots.

Open an issue report revision in Teamcenter Lifecycle Visualization

You can send an issue or an issue report revision, along with an associated visualization dataset, from Active Workspace to Teamcenter Lifecycle Visualization.

1. On the Home page, in the **Search** bar, enter the name of the issue report revision you are looking for and click **Search** .
2. Select it from the search results and click **Open** .
3. On the primary toolbar, click **Open**  > **Open in Visualization** .

Note:

This feature is available only when using Active Workspace 6.1 or later *and* Teamcenter Lifecycle Visualization 14.1 or later.

22. Working with image captures

Capture an image and the model position



An image is a 2D picture or snapshot of a structure (assembly) or a component in a structure. You capture an image to get the view of your structure exactly as it is displayed in the 3D viewer, including all visible markups. Because *image capture* is an independent Teamcenter object, if the original structure is deleted or no longer available, the captured image would still be available.

You can preview images, share them with other users using the *Discussion* feature in Active Workspace, or submit them to workflows.


You can capture an image and the position of a 3D model. You can later view the image capture and work with it in various ways.

Note:

If you have a Base Visualization license, you cannot capture an image using the *Image Capture* feature. However, you can view the images created by other users.

1. **Open the assembly** (product structure) and select the 3D viewer. If you want to associate the image capture with a component of an assembly, select the component.
2. From the work area toolbar, click **Gallery** .
3. In the **Images** tab of the **Gallery** panel, click **Capture Image** .

Note:

The 3D trihedron () is not included in image captures.

4. By default, the image gets a system-generated name and description. You can edit this information to enter the new **Name** and **Description** and click **Add**.

The image displays in the **Images** section of the **Gallery** panel.

5. To capture another image from the **Gallery**, click **Capture Image**.

Tip:

You can **add freehand markups** to image captures.





View an image capture and restore model position

You can view 2D image captures of 3D models in two different ways:

- **Quick view without opening the associated model**
- **View and markup in context of the associated model**

Quick view without opening the associated model

1. Type all or part of the image capture name in the global **Search** box.
2. Select the capture in the results list.
3. In the **Overview** tab, under the **Preview** section, do any of the following:


To	Do this
Zoom in on the image capture, and pan across it	<p>Use the scroll wheel on your mouse to zoom the image in and out.</p> <p>To zoom in on a particular area on the image, double-click the area. To zoom out, double-click the same area again.</p> <p>When you are zoomed in, you can drag to pan across the image.</p>
Fit the image capture to the screen	<p>When zoomed in, double-click.</p>
Add and print markups	<p>From the work area toolbar do any of the following:</p> <ul style="list-style-type: none"> • Click Show Markups  to toggle markups on and off. • Click Markup Panel  to select the markup you want to view. • Click Highlight Markup  and select the required document text. • Click Freehand Markup  to add freehand markups. • Click Markup Shapes to draw a predefined shape of your choice on the desired location of the part or model. • Click Stamp Panel to drag the stamp that you want to include to a specific area on the model. • Click Print Markups to print the markups that you want.

View and markup in context of the associated model

You can view an image capture within its model context.

Note:

- If you have a Base Visualization license, you cannot capture an image using the *Image Capture* feature. However, you can view the images created by other users.
- Images captured while an assembly component is selected are associated with that component. If the component is opened by itself or in a different assembly context than when an image was captured, and the image capture is viewed in the new context, restoration of the model position may look different than expected.

1. Open a model that has an associated image capture.
2. **(Optional)** If the model is an assembly, select an occurrence of a component to set the component as the context for associated image captures.
3. **Load the model** in the 3D viewer.
4. From the work area toolbar, click **Gallery** .
5. In the **Gallery** panel, click **Images**.

The image captures associated with the model (or selected component) are displayed in the **Images** section of the **Gallery** panel.

6. To work with image captures, do any of the following.

To	Do this
Filter the list of image captures	Type a search keyword in the Filter box.

To **Do this**

▼ **Images**



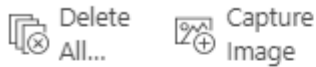


Capture For vlp_vise_side
 Capture For vlp_vise
 Owner: vismockupuser2 (vismockupuser2)

View an image capture in the 3D viewer
 Select the image capture thumbnail from the **Images** tab in the **Gallery** panel.
 To return to viewing the 3D model, click the thumbnail again.

Download the image capture file
 Hover on the desired image in the list and click **Open**

▼ **Images**





Capture For vlp_vise_side
 Capture For vlp_vise
 Owner: vismockupuser2 (vismockupuser2)



Open




Capture For vlp_vise_1
 Capture For vlp_vise
 Owner: vismockupuser2 (vismockupuser2)

View properties of the captured image.








Capture For vlp_vise_1
 Capture For vlp_vise
 Owner: vismockupuser2 (vismockupuser2)

To	Do this
	In the Preview section, open the list associated with the image capture and click Download File  .

▼ Preview




Capture For vlp_vise_side ▼

-  Open
-  Download File
-  Open in New Window
-  Open in New Tab

Delete the image capture	On the Images tab in the Gallery panel, select the image in the list, and click Delete  . Delete appears only if you are logged in as the image owner.
--------------------------	--

Zoom in on the image capture, and pan across it	In the 3D viewer, use the scroll wheel on your mouse to zoom the image in and out. To zoom to a particular area on the image, double-click the area. When you are zoomed in, you can drag to pan across the image.
---	---

Fit the image capture to the screen	When zoomed in, double-click or double-tap.
-------------------------------------	---

View and add markups	<p>Add markups from the Gallery panel</p> <ul style="list-style-type: none"> • In the Images tab, select the image. The Markup panel appears on the work area toolbar. • Click Show Markups  to toggle markups on and off. • Click Freehand Markup  to add freehand markups. • Click Markup Panel  to filter the markup you want to view. You can filter the markups by the page on which the markup occurs, a specific user, the date when the markup was created, or the status of the markup.
-----------------------------	---

7. To close the image and return to the model, in the **Images** list, click the selected image.

23. Working with product and manufacturing information

Overview of product and manufacturing information

You can use the 3D viewer to display the product and manufacturing information (PMI) for a product when the PMI is generated by a CAD program and translated to a JT file. You can view the PMI for a model using either server-side rendering (SSR) or client-side rendering (CSR) modes.

Note:

In the CSR mode, when you select PMI in the **3D** view, it is highlighted in the **3D** view and also in the **Model Views** and **Types** tabs of the **PMI** panel.

In the SSR mode, you can select PMI listed in the **Model Views** and **Types** tabs, but you cannot select PMI by clicking it in the **3D** view.

A JT file can include several different types of PMI, such as dimensions, notes, and so on. You can organize how the PMI is displayed in several different ways. For example, you may need to display only the measurement PMI, or just specific notes about the product. Or, you may want to examine specific sections of the assembly by using model views.

View PMI for a model in the 3D viewer

Perform the following steps to display the product and manufacturing information (PMI) saved within a model in the 3D viewer:

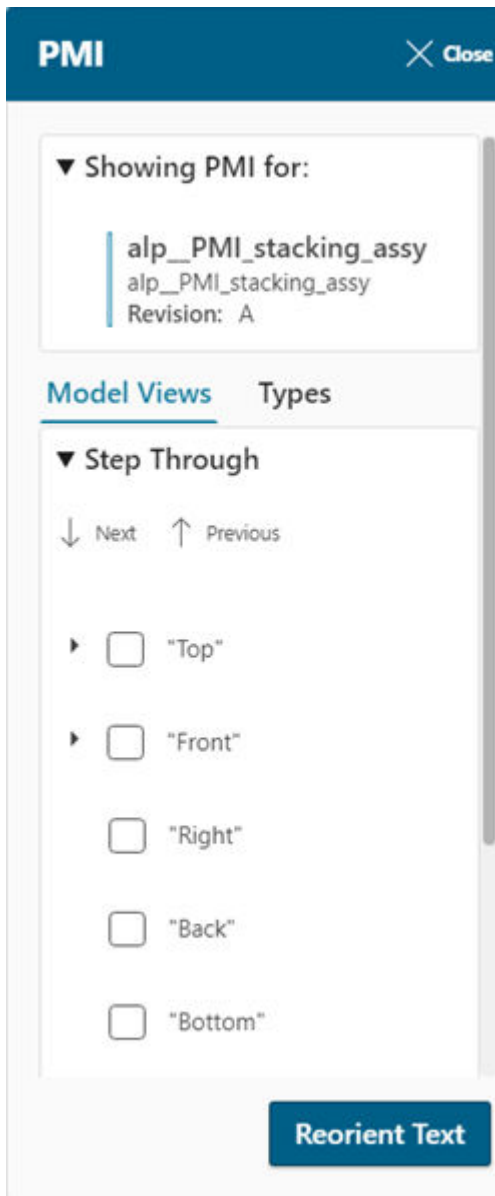
1. **Load a part or an assembly** containing a **DirectModel** dataset that includes PMI, in the 3D viewer.

Note:

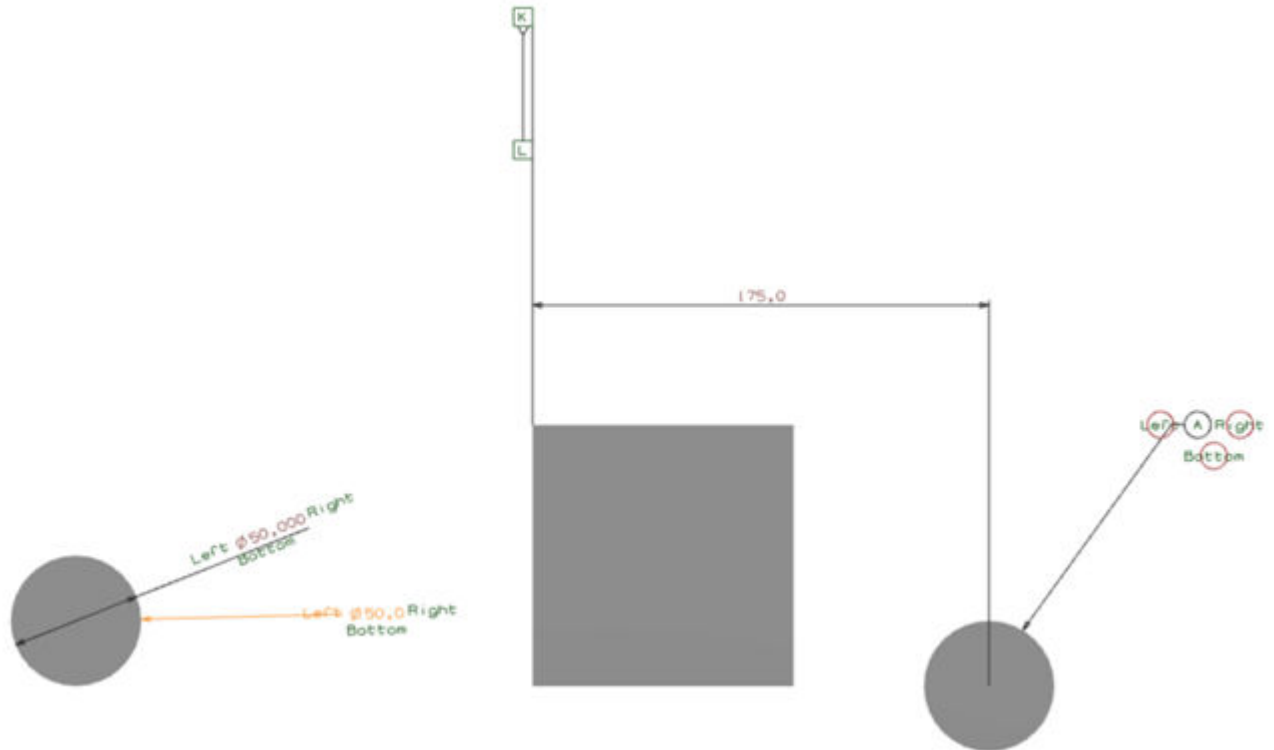
- You can only view the PMI for an assembly if it is available in the assembly-level JT file.
- The 3D viewer does not support displaying PMI data on a leaf node for an assembly.

2. In the 3D viewer, on the work area toolbar, click **More Commands**  -> **PMI** .

The **PMI** panel is displayed and the **Showing PMI for** section displays PMI for the assembly or part that is currently selected.



3. Select the model view or type of PMI you want to see. Based on your selection, the PMI is displayed in the 3D viewer.



Note that PMI is not displayed for items that are hidden. If you select a hidden part, the **Showing PMI for** section in the PMI panel indicates that the part is not currently visible. Click **Show** to unhide the part and see the PMI associated with it.

Moreover, if you select a part that doesn't have any PMI associated with it, the PMI panel displays a message to indicate so.

Work with model views and PMI types

A JT file can include several different types of product and manufacturing information (PMI), such as dimensions, notes, and so on. You can organize how the PMI is displayed in the **3D** viewer using any of the following options on the **PMI** panel:

- **Model Views**
- **Types**



Note:

In the client-side rendering mode, when you select PMI in the **3D** view, it is highlighted in the **3D** view and also in the **Model Views** and **Types** tabs of the **PMI** panel.

In the server-side rendering mode, you can select PMI listed in the **Model Views** and **Types** tabs, but you cannot select PMI by clicking it in the **3D** view.

To work with model views or PMI types in the viewer, do any of the following in the **PMI panel**.

Action	How to do this
Display a list of model views or PMI types present in the model	<p>Depending on whether you want to view model views or PMI types, click the Model Views or Types tab.</p> <p>The PMI visibility state in the viewer is maintained when you switch between the Model Views and Types lists.</p>
Apply or remove a model view	<ol style="list-style-type: none"> <li data-bbox="488 642 1425 701">Apply a model view from the Model Views list by selecting its check box. <div data-bbox="573 730 1450 1003" style="border: 1px solid black; padding: 10px; margin: 10px 0;"> <p>Note:</p> <ul style="list-style-type: none"> <li data-bbox="594 806 1385 869">• You can also apply or remove a model view by stepping through the list using the Next ↓ and Previous ↑ buttons. <li data-bbox="594 915 1398 978">• You can step through full model views only, and not through the individual PMI types contained within model views. </div> <ul style="list-style-type: none"> <li data-bbox="557 1050 1463 1146">• If you apply a model view that consists of multiple PMI types, all PMI types within the selected model view also get applied and are shown in the viewer. <li data-bbox="557 1192 1414 1255">• Expand the model view to view the different PMI types contained within it. <li data-bbox="557 1302 1463 1398">• You can choose which PMI type should be shown or hidden by selecting or clearing the check boxes from the list. If you clear all PMI types within a model view, the model view itself will still be retained. <li data-bbox="557 1444 1414 1478">• Collapse the model view to hide all PMI types contained within it. <div data-bbox="508 1503 1450 1633" style="border: 1px solid black; padding: 10px; margin: 10px 0;"> <p>Note:</p> <p>At any particular instant, only one model view can be applied.</p> </div>
Show or hide PMI types	<ol style="list-style-type: none"> <li data-bbox="488 1682 1377 1711">Show a PMI type in the viewer by selecting it from the Types list.

Action	How to do this
	<div data-bbox="573 241 1448 447" style="border: 1px solid black; padding: 5px;"> <p>Tip:</p> <p>You can also show or hide a PMI type by stepping through the list using the Next ↓ and Previous ↑ buttons. These buttons enable you to step through each type within each group.</p> </div> <ul style="list-style-type: none"> • If you select a PMI group that consists of multiple types, all PMI types within the selected group are also shown in the viewer. • Expand the group to view the different PMI types contained within it. • You can choose which PMI type should be shown or hidden by selecting or clearing the check boxes from the list. If you clear one type from the group, the group also gets cleared. However, the other types within that group remain selected. • Collapse the group to hide all PMI types contained within it. • To hide PMI, select a PMI in the viewer > access the context menu by right-clicking the PMI > click Hide PMI . <div data-bbox="506 1020 1448 1186" style="border: 1px solid black; padding: 5px;"> <p>Note:</p> <p>At any particular instant, more than one PMI groups or types can be shown.</p> </div>
Highlight PMI types	Select the PMI type by clicking on it to highlight it in the viewer. Clear the selection to reverse the action.
Show the PMI for a different part or reset the PMI for the current part	<p>Select the part in the assembly.</p> <p>If you show the PMI for a part and clear it or select another part, the PMI for the previously selected part remains visible. To hide the PMI for any previous part, navigate back to that part and hide its PMI.</p> <p>If you clear a part, the selection reverts back to the last subassembly displayed in the breadcrumb.</p>
Reorient the PMI text	When you rotate the view in the viewer, the PMI annotations rotates, too, and becomes unreadable. To reorient the PMI annotations so that they become readable, click Reorient Text at the bottom of the PMI panel.
Display PMI annotations in the plane of the screen	<p>To lock the PMI annotations so that they do not rotate with the model in the viewer and remain readable.</p> <ol style="list-style-type: none"> 1. From the primary toolbar, click Viewer Options .

Action

How to do this

-
2. In the **Orientation** section, select the **PMI > Show flat to screen** check box.

To undo the selection, clear the **Show flat to screen** check box.

24. Save filtered and configured structures within a workset

About worksets

A workset is your personal working context. You can save your product definition in a workset. It is a container that holds one or more structures together. These structures can be the different views of the same structure or two entirely different structures. You can perform all BOM operations for a structure within a workset even when a partition scheme is applied to the structure.

You can create and use worksets only if your Teamcenter setup has the Smart Discovery license.

Create a snapshot of a workset

You can create a snapshot of all worksets, irrespective of whether a partition scheme applied to the structures within the worksets.

Snapshots capture the 3D data associated with your product. When you load the product in the 3D viewer and take a snapshot, you capture the current 3D view, including camera, visibility, selection, view port (pan, zoom, and rotate modes), orientation, sections, measurements, queries, and markups. You also capture the configuration and filtering criteria.

When working with worksets, you can capture snapshots of a workset to preserve the working context and save it for later use.

Working with Product Configurator data in a workset

A workset is a container that holds two or more structures together. Therefore, to view configuration information, select the relevant structure within the workset.

You can perform Product Configurator-related tasks, such as authoring variant conditions and loading a Saved Variant Rule (SVR), from within the workset and from the structures included in the workset. Classic variants are not supported in a workset.

Modifying the data in a workset

After you open a workset, it is always loaded with static data. You can modify the data in a workset, for example, you can add a structure element, remove an element, and edit the properties of an element. When you do this, you must replay the workset or a structure within the workset for the changes to take effect.

When you insert an element in a structure within a workset, the element is immediately displayed in the tree structure. However, when you reopen the workset or refresh the browser page, the element is not

displayed. Now, when you replay the workset or a structure within the workset, the element might be displayed in the tree structure based on the applied filter criteria.

When you delete an element from a structure within a workset, the element is removed from the tree structure.

When you edit the properties of an element or edit a variant condition or author occurrence effectivity, the updates are immediately done. However, to display the changes, you must replay the workset or the structure within the workset. Now, when you reopen the workset or refresh the browser page, the updates are maintained even though the changed property does not satisfy the filtering or configuration criteria.

Export and import structures along with worksets

You can export and import structures along with worksets to and from other Teamcenter sites.

A workset is a context collector for other structures. If your organization has very large structures with a multitude of occurrences, these occurrences can be collated within a workset to do a *what-if* analysis. But when you send the workset to another site by using a Multi-Site environment or a Briefcase, you might not want to export such a large number of occurrences along with the workset. Therefore, while exporting, the **Include entire BOM** option is not enabled by default. Consequently, the TCXML-based Multi-Site functionality works to share workset and related objects but not the product item or item revisions or their content.

Worksets also support site-consolidation activities through the TCXML-based Multi-Site functionality.

To export and import structures along with worksets, you can use:

- Briefcase files
- Multi-Site Collaboration
- PLM XML

25. Working with Clearance results

About Clearance

What is Clearance analysis?

Clearance analysis identifies proximity violations between elements or groups in your 3D model. Elements could be parts, leaf components, or end items.

The clearance requirement is the minimum allowed distance between any two elements within the analysis scope.

What is ClearanceDB?

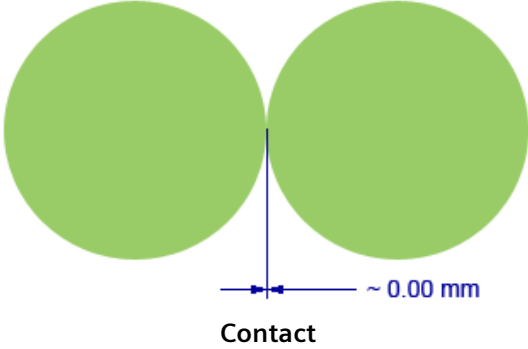
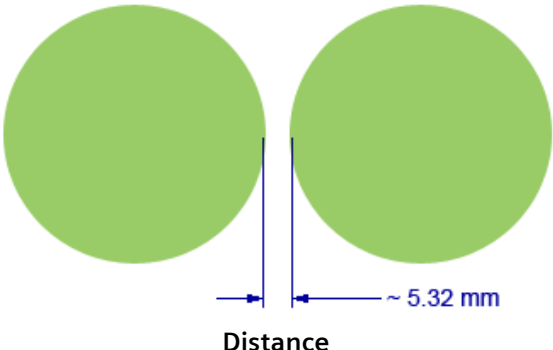
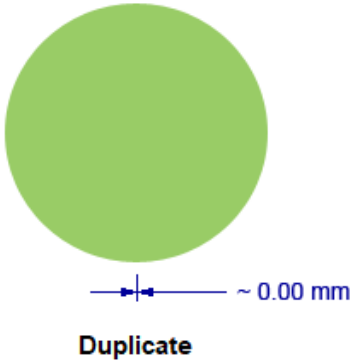
ClearanceDB is a process and an associated set of tools that enables customers to detect, manage, and resolve spatial interference issues as they arise during a product's lifecycle. By automating this process and constantly re-evaluating on a regular basis, customers can identify design flaws that may have otherwise delayed the release of new product offerings, increased costs, or jeopardized quality.

Types of Clearance results

When a clearance check occurs in the Teamcenter Clearance Database (ClearanceDB), any two parts or elements that are closer to one another than a specified distance are identified as an issue for further investigation. You can [view the results](#) of a clearance check in Active Workspace.

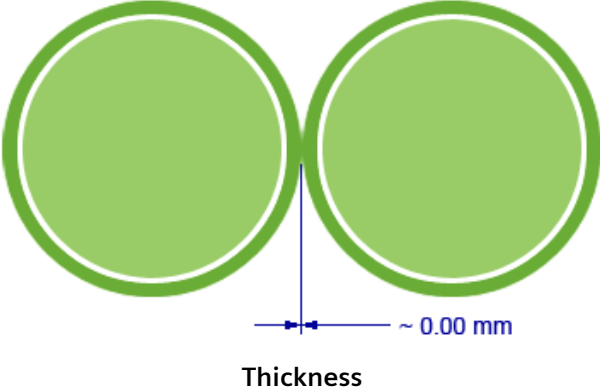
The following table describes each *Clearance* type and provides examples to help you understand the criteria used to identify them.

Clearance type	Appears in the results list as	Description and example
Contact	C	<p>A hard interference. The elements are in contact but do not penetrate. The distance result is always ~ 0.0.</p> <div style="border: 1px solid black; padding: 10px;"><p>Note:</p><p>Clearance results are approximate if your elements do not consist of precise geometry.</p><p>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.</p></div>

Clearance type	Appears in the results list as	Description and example
		 <p style="text-align: center;">Contact</p>
Distance	D	<p>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.</p>  <p style="text-align: center;">Distance</p>
Duplicate	Dup	<p>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.</p>  <p style="text-align: center;">Duplicate</p>
Failure	F	<p>The Clearance calculation has failed to complete. This may happen if the Clearance calculator crashes.</p>

Clearance type	Appears in the results list as	Description and example
Penetration	P	<p>A hard interference. One element penetrates or is completely contained by another. The distance result is always a value less than ~ 0.0.</p> <div data-bbox="487 424 1453 588" style="border: 1px solid black; padding: 5px;"> <p>Note: Penetration values are always approximate, even when using precise geometry.</p> </div> <div data-bbox="719 638 1214 949" style="text-align: center;"> </div> <p style="text-align: center;">Partial penetration</p> <p>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.</p> <p>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.</p> <p>If the minimum distance between elements is ~ 0.0 and the parts do not contain NURBS data, the issue is classified as a contact.</p> <div data-bbox="745 1476 1235 1774" style="text-align: center;"> </div> <p style="text-align: center;">Full penetration (elements in contact)</p> <p>If the minimum distance between elements is greater than ~ 0.0 (the elements are not in contact with one another), the penetration depth is the</p>

Clearance type	Appears in the results list as	Description and example
		<p>minimum distance plus the distance the minimum distance vector extends to the further side of the inner elements' bounding box.</p> <div data-bbox="730 451 1250 756" style="text-align: center;"> </div> <p style="text-align: center;">Full penetration (elements are apart)</p> <div data-bbox="487 850 1453 1312" style="border: 1px solid black; padding: 10px;"> <p>Note:</p> <ul style="list-style-type: none"> • 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. </div>
Thickness	T	<p>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.</p> <div data-bbox="487 1501 1453 1732" style="border: 1px solid black; padding: 10px;"> <p>Note:</p> <p>Element pairs with material thickness values that come into contact or penetrate one another are identified as the <i>Unknown</i> type. Contact, penetration, and distance line analysis are not available for Clearance issues that contain element pairs with material thickness values.</p> </div>

Clearance type	Appears in the results list as	Description and example
		
Unknown	U	A hard interference. Further analysis is required to classify as a contact or penetration.

Tasks to perform with Clearance results

You can perform any of the following actions with Clearance results:

- **View and analyze** Clearance results in the 3D viewer.
- **Disposition** a result to specify details for each Clearance result, capturing information, such as the owner, priority, status, and so on.
- **Display** a cross section at regions of contact or penetration for a selected Clearance result.
- **View** clearance result images to help relevant stakeholders to understand the issue.
- **Search** the results list for specific elements, for example, a part name.
- **Filter** the results list to see the data that is relevant to you.
- Customize the results list to reorder, hide, or display columns in the results table.
- **Define** Clearance preferences to change the annotation displays for the results.

View and analyze Clearance results

You can view the results of a *Clearance* analysis in Active Workspace. The Clearance results display the parts with a Clearance issue, the distance value between part pair, and other relevant information such as the interference type.

Note:

You can only see Clearance results if the Clearance database has been correctly configured for Active Workspace by your administrator and there are results for the selected model in the database.

Procedure

1. **Load the model** in the 3D viewer.
2. On the work area toolbar, click **Clearance Results**  to load the Clearance results list for the model in the viewer.

The Clearance results are displayed as an *analysis summary* in the viewing window and as a results list below the viewing window.

3. Analyze Clearance results in the 3D viewer.


The *analysis summary* specifies amongst other things, the number of parts that were analyzed, the number of results that were found for parts which were penetrations, contacts, distances, and duplicates.

Matrix Clearance Analysis Summary







Elements Analyzed: 48
 Total Results: 80
 Visible Results: 80 of 80

Penetrations (P): 44
 Contacts (C): 26
 Distances (D): 10
 Duplicates (Dup): 0

Element Type: PART
 Requirement: 0.2



Analysis Mode | Cross Section | Image Gallery | Disposition | Reset View | Refresh Table | Analysis Pair... | Intersection Fit | Search Results


Icon	Type	Element Number 1	Element Number 2	Result
	C	HDD-0523/A;1-Magnet Frame Lower	HDD-0552/A;1-Magnet	~ 0.0000
	C	HDD-0523/A;1-Magnet Frame Lower	HDD-0554/A;1-Base Plate	~ 0.0000
	P	HDD-0523/A;1-Magnet Frame Lower	HDD-0556/A;1-Magnet Frame Upper	~ -1.0000
	D	HDD-0502/A;1-Lock Gimbal	HDD-0541/A;1-Lock Actuator Gimbal	~ 0.0053
	P	HDD-0502/A;1-Lock Gimbal	HDD-0554/A;1-Base Plate	~ 0.0000
	P	HDD-0502/A;1-Lock Gimbal	HDD-0532/A;1-Cover	~ -0.0706

4. If you are in the **analysis mode**:

- If you select a particular row in the results table, the summary in the 3D viewer is updated to provide details only for the element (part) pairs in the selected row.
- The relevant parts in the 3D viewer display annotations where the issue occurs.
- If you have selected **Measurement Annotation** in the **Preferences** panel, on selecting a particular row in the results table, the relevant parts in the 3D viewer display measurement annotations based on the type of the Clearance issue.

For example, if the type of Clearance issue is a penetration, then measurement annotations specify how big the penetration is between the parts. This can help a reviewer understand the type of details to add when they disposition a result.


You can also see contact or penetration points, an on-screen analysis summary, and a penetration vector in analysis mode. These can all be turned on and off in the **Preferences** panel.




- If required, you can **view a cross section** of a violating part.
- (Optional) To reorder, hide, or display columns in the results table, click **Table Settings**  > **Arrange**.

Disposition a Clearance result

You can specify details for each Clearance result, capturing information, such as the owner, priority, status, and so on. You can then display this issue *disposition* information in the clearance results list. This makes filtering easier for reviewers who may only wish to see issues with a specific owner or only those issues that are marked important. As reviewers investigate each result, they can reassign priorities. This disposition information gets saved in the Clearance database.

Procedure

- View the Clearance results** for the model.
- In the results list, either select a single result or select the multiple results for which you wish to add details.
- On the Clearance results toolbar, click **Disposition** .
- In the **Disposition** panel, specify the relevant details.

Option	Description
Owner	<p>Provide the name of the owner of the contact or penetration issue.</p> <ol style="list-style-type: none"> Click Add Owner . <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Note:</p> <p>Add Owner  is enabled when the owner field is not populated. If an owner has been assigned, remove the owner before adding a new one.</p> </div> <ol style="list-style-type: none"> From the Disposition panel, use the search box to quickly find the desired owner. (Optional) Click the Filter  icon for additional search filters. <p>Filters are defaulted to Type, Group, or Role. Select one of the following options to show which users display in the list.</p> <ul style="list-style-type: none"> • Under Type, select Group Member to show all the users in your group.


Option	Description
	<ul style="list-style-type: none"> Specify group members by Group or Role. Enter a name in the specific search box to filter. <p>d. Click Add.</p>
Priority	Choose the priority of the contact or penetration issue from the following options: <ul style="list-style-type: none"> None Low Medium High
Zone	Choose the location of the issue, for example, the right wing.
Status	Choose the status of the contact or penetration issue from the following options: <ul style="list-style-type: none"> Not Evaluated Not Relevant Relevant Fixed
Important	Select the check box if the issue is important and needs attention.
Comment	Enter additional details about the issue.
Change Required	Select the check box if the issue requires a change.
Change Reference	If you selected the Change Required check box, enter additional information about the change.


- Click **Apply**.

View a cross section of a violating part

You can view cross section planes at regions of contact or penetration for a selected Clearance result.

Procedure

- View the Clearance results** for the model.
- On the Clearance results toolbar, click **Analysis Mode** .

3. Select a result from the results list in the viewer.
4. On the Clearance results toolbar, click **Cross Section** .

Note:

Cross Section is enabled only after you select **Analysis Mode**.

A cross section view of the selected result is displayed in the viewer. This helps you get a good view of what the problem is.

View images for Clearance results


In Active Workspace, you can view images of each result in the Clearance results **Image Gallery**.

These images are captured by Teamcenter lifecycle visualization mockup and stored in the Clearance database for each result. The images are 2D jpeg images associated with each result.

Note:

You can only see 2D images for Clearance results if they have been captured by your administrator and uploaded into the Clearance database.

Procedure

1. **View the clearance results** for the model.
2. Select a result from the results list in the viewer.
3. On the Clearance results toolbar, click **Image Gallery** .
4. From the **Image Gallery** that opens, for the selected result, select the image to display.

This 2D image is a visual display that helps the relevant stakeholders to understand the issue.



Search the Clearance results list

You can filter the Clearance results list by using a search term, such as a part name. This is particularly useful if you have a large results list and need to focus on only the results for a specific element.

Procedure

1. **Open a model** in the 3D viewer.
2. **View the Clearance results** for the model.

Type	Part Number1	Part Number2	Result
P	HDD-0517/A;1-PCB3 Motor Drive	HDD-0526/A;1-Connector	~ -0.0003
P	HDD-0541/A;1-Lock Actuator Gimbal	HDD-0554/A;1-Base Plate	~ -0.0020
P	HDD-0510/A;1-Platter	HDD-0536/A;1-Retainer	~ -0.0045
P	HDD-0547/A;1-Gasket Feedthrough	HDD-0520/A;1-Connector Feedthrough	~ -0.0059
P	HDD-0534/A;1-Nut Spindle	HDD-0519/A;1-Gimbal Hub	~ -0.0142
P	HDD-0545/A;1-Motor Shaft	HDD-0542/A;1-Motor Core	~ -0.0153

- On the Clearance results toolbar, click **Search Results** .
- In the **Search Results List** dialog box, enter the name of the part that you want to search for, for example, *Nut*.
- (Optional) Select the **Case Sensitive** check box to create string searches that are case sensitive.
- Click Search .

The results list displays only those results that match the value entered in the **Search Results List** dialog box.

Type	Part Number1	Part Number2	Result
C	HDD-0509/A;1-Gimbal Armature	HDD-0534/A;1-Nut Spindle	~ 0.0000
P	HDD-0534/A;1-Nut Spindle	HDD-0519/A;1-Gimbal Hub	~ -0.0142

Filter the Clearance results list


You can filter the data within a table column so you can easily see the data that is important to you.

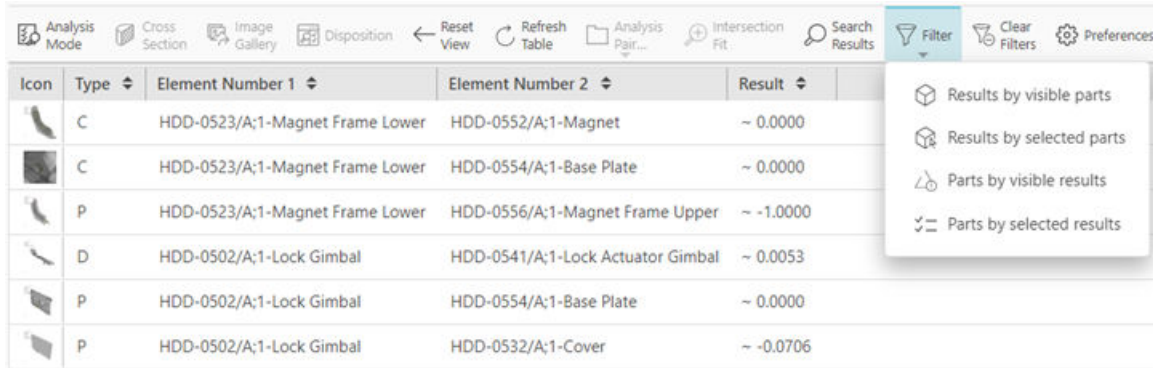
- View the Clearance results** for a model.
- Click a column heading to access available filters.







Both the analysis summary in the viewing window and the results table are updated to show a filtered list of Clearance issues.

Sync the Clearance results list and the 3D view

Use this technique to sync the Clearance results list with the parts that are currently displayed in the viewer or vice versa.

1. **View the clearance results** for a model.
2. On the results toolbar, click **Filter** .



Icon	Type	Element Number 1	Element Number 2	Result
	C	HDD-0523/A;1-Magnet Frame Lower	HDD-0552/A;1-Magnet	~ 0.0000
	C	HDD-0523/A;1-Magnet Frame Lower	HDD-0554/A;1-Base Plate	~ 0.0000
	P	HDD-0523/A;1-Magnet Frame Lower	HDD-0556/A;1-Magnet Frame Upper	~ -1.0000
	D	HDD-0502/A;1-Lock Gimbal	HDD-0541/A;1-Lock Actuator Gimbal	~ 0.0053
	P	HDD-0502/A;1-Lock Gimbal	HDD-0554/A;1-Base Plate	~ 0.0000
	P	HDD-0502/A;1-Lock Gimbal	HDD-0532/A;1-Cover	~ -0.0706

3. Select one of the following options:

- **Results by visible parts**

List the results that are related to all parts visible in the 3D viewer or in the tree view.

- **Results by selected parts**

List the results related to all parts that are selected in the 3D viewer or in the tree view.

- **Parts by visible results**

Display the parts in the 3D viewer that are related to all results that are currently shown in the results table. Use **column filters** to list results for specific parts first.

- **Parts by selected results**


Display the parts in the 3D viewer that are related to all results that are currently selected in the results list.

To clear all filters, click **Clear Filters** .

Define Clearance preferences

Use the Clearance preference settings to change the annotation displays for the Clearance results.

Procedure

1. **View the clearance results** for a model.
2. On the results toolbar, click **Preferences** .
3. Select any of the following parameters.

Use this option	To do this
Contact/penetration points	Display the point of contact or penetration for the selected result.
Contact Point Preferences	Choose the color and size of the contact points specifying the issue.
On-screen analysis summary	<p>Display the summary of the analysis in the 3D viewer, including the following parameters:</p> <ul style="list-style-type: none"> • Elements Analyzed: The total number of elements analyzed. • Total Results: The total number of Clearance results found. • Visible Results: The total number of Clearance results that are available to be displayed in the viewer. • Penetrations: The number of penetrations found during analysis • Contacts: The number of contacts found during analysis. • Distances: The number of distance violations found during analysis. • Duplicates: The number of duplicate parts found during analysis. • Element Type: The element type that was used during analysis: part, leaf component, or end item. • Requirement: The minimum distance that two elements are allowed to be from each other without being in violation. <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p>Tip:</p> <p>A Clearance requirement value of 0 identifies only elements that are in contact with or penetrating other elements. To create distance (D) violations, a requirement greater than 0 must be defined.</p> </div>

Use this option	To do this
	<div style="border: 1px solid black; padding: 10px;"> <p>Note:</p> <p>When you are in analysis mode, if you select a particular row in the results table, the summary in the 3D viewer is updated to provide details only for the element (part) pair in the selected row.</p> </div>
Penetration Vector	<p>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.</p> <p>Select the color of the arrow from the list.</p>
Measurement Annotation	<p>Display the measurement annotation as specified in the Result column for the element (part) pair in the selected row.</p> <p>For the selected row in the results table, the relevant parts in the 3D viewer display measurement annotations based on the type of the Clearance issue. For example, if the type of Clearance issue is a penetration, then measurement annotations specify how big the penetration between the parts is.</p>

26. Clear all added content from the current view

You can remove content such as measurements and sections, reset part positions and colors, and hide PMI in the 3D viewer. Clearing the view retains current part visibility and camera view and does not affect the selection state.




Clearing the 3D viewer has the same effect as individually removing the following user-generated content:

- It resets exploded views.
- It deletes 3D markup.
- It deletes measurements and geometric information.
- It clears any unsaved part positions returning parts to their saved locations.
- It removes Clearance analysis results.
- It resets part color, true shading, and materials and it restores the original color.

Note:

There are no individual controls within the application to alter or restore color and material information.

To remove all added content from an already **opened assembly**:



1. On the toolbar, click **Visibility Controls** .
2. Click **Clean 3D View** . Alternatively, right-click in the viewer, and select **Clean 3D View**  from the context menu.
3. Click **Remove** in the message that is displayed.

Note:

Authored annotations, such as measurements and sections, are deleted by the **Clean 3D View** command, but PMI is hidden, not deleted, as it is persistent content from a JT file.

27. Troubleshoot Visualization

The following table lists some issues, which you may encounter while working with the Visualization of 3D Product Data solution, and their resolutions.

Issue	Possible resolution
<p>In Active Workspace, the current render location is set to client-side rendering (CSR) and you are attempting to either:</p> <ul style="list-style-type: none"> View Massive Model Visualization (MMV) data. View data too large for CSR to load. 	<p>Probable causes — MMV data or very large data is displayed in the viewer only when the render location is set to server-side rendering (SSR).</p> <p>Solution — To resolve this issue, change the render option to SSR using the following method:</p> <ol style="list-style-type: none"> 1. Load a single part or a smaller assembly using CSR. 2. From the work area toolbar, in the Viewer Options panel, under Render Options, change the Model Render Location to Server for SSR rendering. <p>After you change the render option, load the assembly that you were originally trying to view.</p>
<p>Get settings and status information about the Visualization Server process when there a loading issue or a startup issue with the 3D viewer.</p>	<ol style="list-style-type: none"> 1. In Active Workspace, in the 3D viewer, from the work area toolbar, click Viewer Options . 2. On the Viewer Options panel, under Diagnostics, download  the Viewer Diagnostic Report File. <p>You can also use the Performance Monitor command to view Visualization Server performance and other optional telemetry data.</p>
<p>The My Gallery tile is not available on the home page in Active Workspace.</p>	<p>Verify the following:</p> <ul style="list-style-type: none"> • Ensure that your administrator has installed the Active Workspace 3D Visualization module. • Ensure that your administrator has configured the My Gallery tile for your workspace. By default, the My Gallery tile is available for the Author, Consumer, and the Default workspaces.