

Enhancement Details: Creo Illustrate Creo 2.0

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Enhancement Details

Creo Illustrate

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Phantom parts in a figure saved as a 2D illustration file retain their phantom appearance in Arbortext IsoDraw.

Translate and Rotate Sections with Draggers

You can translate and rotate sections with draggers.

Unscrew Coupled Parts in Animations

There is an Unscrew dialog box.

Validating an Illustration

You can validate an illustration by comparing the contents of an exported companion XML file to the sBOM, figures, and item lists in the .c3di file.

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Visualization

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Enhancement Details

3D Illustration sBOM Exports as 3D PDF Model Tree

You can export a 3D Illustration sBOM as a 3D PDF model tree.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click File > Save Figure As > As a 3D file. In the Type box select PDF (*.pdf).
Processes, Initiatives, and Best Practices	

Benefits and Description

Creo Illustrates saves the sBOM to the Adobe Acrobat Model Tree when you save a figure as a 3D .pdf file. When you open the exported 3D PDF file and click Toggle Model Tree, the sBOM tree opens in the Model Tree navigation panel.

Enhancement Details

Add Construction Geometry to Figures

You can add construction geometry to figures.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Tools and then click Point, Axis, Plane, Coordinate System, Curve or Line.
Processes, Initiatives, and Best Practices	

Benefits and Description

Use the **Point**, **Axis**, **Plane**, **Coordinate System**, **Curve**, and **Line** tools to add construction geometry to figures in the figure window. Construction annotations appear in the View Content pane of the upper data panel. Like other annotations, constructions are associated and saved with the figure you create them in. You can also save and load construction geometry.

Enhancement Details

Add PNG Symbols

You can add .png format files to the Symbol Library.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	
Processes, Initiatives, and Best Practices	

Benefits and Description

Symbols added from .png files retain .png image capabilities, such as multiple transparency levels, gamma correction, and interlacing.

Enhancement Details

Cap Section Cuts for Non-Solid Parts

You can cap section cuts for non-solid parts.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Select non-solid parts, right-click the parts in the figure, and then select Edit Properties. In the Item(s) Properties dialog box, click appearance, and then select Treat as Solid Geometry.
Processes, Initiatives, and Best Practices	

Benefits and Description

By default, Creo Illustrate only caps section cuts for solid parts when Cap Section is selected. Select Treat as Solid Geometry to cap section cuts for non-solid parts.

Enhancement Details

Combining sBOM Nodes

Combining sBOM Nodes is enhanced.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	In Structure Edit mode click Home > Combine.
Processes, Initiatives, and Best Practices	

Benefits and Description

You can combine selected child sBOM nodes with their parent, whether the parent is selected or not. If you select child nodes and their parent node, the child nodes combine with the parent, and the parent node combines with its parent one level up. You can also right-click a parent node and select Combine All Children to combine all of its children.

Enhancement Details

Copy Item List to Clipboard

You can copy one or more rows from the item list to the clipboard.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	In the lower data panel Item List tab, select one item list row or all rows (CTRL+A). Right-click the item list, and then click Copy Selected.
Processes, Initiatives, and Best Practices	

Benefits and Description

From the item list, you can copy all or selected rows along with the item list column heading, to the clipboard. Paste the item list text into a spreadsheet text editor. The text is tab-separated.

Enhancement Details

Display 3D Illustration Item Lists in Arbortext Editor

You can display 3D Illustration Item Lists in Arbortext Editor.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	
Processes, Initiatives, and Best Practices	

Benefits and Description

A new application program interface enables Arbortext Editor to extract item lists from published Creo Illustrate .pvz files and display them.

Enhancement Details

Drag a Selection Rectangle to Select Callouts and Notes

You can drag a selection rectangle to select callouts and notes.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	
Processes, Initiatives, and Best Practices	

Benefits and Description

You can drag a selection rectangle around one or more callouts or notes in the figure window. Release the mouse button to select any callouts or notes that are fully or partially inside the selection rectangle.

Use the **Markups** selection method to select only callouts and notes in the selection rectangle. To prevent the selection of callouts and notes in the selection rectangle, use a selection method other than **Markups** or **Smart**.

Enhancement Details

eBOM and sBOM Tabs and Parts Tab Open

The eBOM and sBOM tabs open along with the Parts tab.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	
Processes, Initiatives, and Best Practices	

Benefits and Description

When you select a part in the eBOM or sBOM, the corresponding eBOM or sBOM tab opens together with the Parts tab. If you select a part in the figure window, the eBOM tab opens with the Parts tab.

Enhancement Details

Edit 3D Symbol Thumbnails Interactively

You can edit 3D Symbol Thumbnails Interactively.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Home > Symbols > More Symbols, and then click Manage.
Processes, Initiatives, and Best Practices	

Benefits and Description

You can interactively adjust the appearance of 3D symbol thumbnail images in the Symbol Library. From the Creo Illustrate Options dialog box, you can edit thumbnails for new or existing 3D symbols. From the Add to Library and Edit Symbol dialog boxes you can perform the actions listed below:

- Spin, zoom, and pan the view
- Select a predefined orientation
- Select the thumbnail image scale and coordinate system

Enhancement Details

Edit Attributes for sBOM Parts

You can edit attributes for sBOM parts.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Select the sBOM tree nodes and click sBOM > Edit Properties. In the Item(s) Properties dialog box, click attributes.
Processes, Initiatives, and Best Practices	

Benefits and Description

Add, delete, or change the values of attributes for selected sBOM parts in Structure Edit mode. Editing sBOM attributes does affect eBOM attributes. However, edited sBOM attributes override their corresponding source eBOM attributes. You can review edits made to selected sBOM part attributes in the lower data panel Attributes pane.

Enhancement Details

Find and Highlight Parts in the sBOM and eBOM

Parts matching a search criteria are highlighted in the sBOM and eBOM.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	In Structure Edit mode, select Home > Find. In the Find dialog box, type the part name or attribute value to search for. Next, select the search scope.
Processes, Initiatives, and Best Practices	

Benefits and Description

Use **Find** in Structure Edit mode to search for parts by part name and attribute value. Parts that match the search criteria are highlighted in the sBOM, eBOM, and in the Structure Edit window

Enhancement Details

Items Are Highlighted in the Item Lists and the sBOM

When you select annotations, associated items are highlighted.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	To change the highlight color, click File > Options, and then click Global > General > Navigation. On the right, change the Secondary selection color, and then click Apply or OK.
Processes, Initiatives, and Best Practices	

Benefits and Description

Selecting annotations such as callouts or measurements in the Figure window highlights their associated items in the Item List and sBOM. You can change the color of the highlighting.

Enhancement Details

Measure Parts in Figures

You can measure the dimensions of parts and part features.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Tools and then click Distance, Diameter, Radius, or Angle.
Processes, Initiatives, and Best Practices	

Benefits and Description

You can use the **Distance**, **Diameter**, **Radius**, and **Angle** tools to measure the dimensions of parts and part features in the figure window. Measurement annotations appear in the **View Content** pane of the upper data panel. Like other annotations, measurements are associated and saved with the figure you create them in.

Enhancement Details

Protect Intellectual Property (IP) in Published Illustrations

Creo Illustrate offers IP protection for published illustrations so you can distribute them with less risk.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Publish > Settings and then select Include referenced parts only or Configuration recipe. To add a recipe to the Configuration list, open the Creo Illustrate Options dialog box, and then select Global > Publishing > 3D > Add.
Processes, Initiatives, and Best Practices	

Benefits and Description

Use the options listed below to enable and change IP protection settings:

- Include referenced parts only—When selected, the published illustration .pvz file only includes illustrated parts. Parts that are not illustrated are omitted.
- Configuration recipe—Select the name of an IP protection publishing recipe from the Configuration list.

Enhancement Details

Publish as 2D

You can set Creo Illustrate for 2D or 3D publishing.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Publish > Settings and then select the 2D illustration file format: CGM, IsoDraw, or SVG. Click Manage Settings to view or change 2D HLR options.
Processes, Initiatives, and Best Practices	

Benefits and Description

2D publishing creates a 2D HLR (hidden lines removed) illustration view in the published .pvz file for every figure in the .c3di file.

Enhancement Details

Recover Illustration from Automatically Saved File

You can recover an illustration from an automatically saved file.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	The Recover Files dialog box.
Processes, Initiatives, and Best Practices	

Benefits and Description

Creo Illustrate periodically saves your illustration as you make changes. You can see messages about these automatic saves in the Messages pane on the lower panel.

If Creo Illustrate exits prematurely before you save changes, the Recover Files dialog box opens when you restart the application. Options for recovering files are listed below:

- Recover Selected—Recovers your file using the currently selected backup file in the list. Save the illustration file after Creo Illustrate recovers it.
- Delete All—Deletes all backup files in this list. You cannot recover after you click Delete All.
- Skip for now—Enables you to recover the file later. After you click Skip for now, Creo Illustrate opens a new file. Click File > Recover Files later to reopen the Recover Files dialog box and recover your file

Enhancement Details

Routing Explode Lines through Intermediate Parts

You can route explode lines through intermediate parts.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Select the explode line, right-click, and select Insert Intermediate Point. In the Select Intermediate Point dialog box, click the Reference box.
Processes, Initiatives, and Best Practices	

Benefits and Description

You can route an existing explode line through an intermediate point on a part or part feature.

Enhancement Details

Save Figure As SVG Illustration File

You can save a figure as an SVG Illustration file.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click File > Save Figure As > As an illustration file. In the Save Figure As Illustration dialog box, click the Type box, and then click SVG (*.svg). Click Save.
Processes, Initiatives, and Best Practices	

Benefits and Description

From the Save Figure As Illustration dialog box, you can save a figure as a 2D illustration file in .svg format.

Enhancement Details

Set Up an Illustration Automatically

You can automatically set up an illustration by importing a companion XML File.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Import a CAD file and then click Home > Import.
Processes, Initiatives, and Best Practices	

Benefits and Description

After you import a CAD file, instead of manually creating an sBOM in Structure Edit mode, you can import a companion XML file. The XML file contains <sbom>, <figure>, and <itemlist> elements. Creo Illustrates uses these elements to automatically perform the illustration setup tasks listed below:

- Map nodes in the eBOM to single or combined nodes in the imported sBOM.
- Apply custom part names and properties to sBOM nodes while preserving eBOM links.
- Apply custom ITM and Item Name values to item list while preserving sBOM links.
- Generate a set of named empty figures with populated item lists.
- Define sBOM parts as itemized or not. Itemized parts have item list entries. Parts that are not itemized do not.

Enhancement Details

Setting the Spin Center

You can set the spin center relative to part geometry.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Click Spin Center > Automatic Spin Center.
Processes, Initiatives, and Best Practices	

Benefits and Description

Select Automatic Spin Center and then use one of the methods listed below to set center points for spinning:

- Select a point on a part to make that point the spin center.
- Select other part geometry, such as a face, an edge, or the whole part, to set the spin center at the center of that geometry.
- Press and hold ALT when selecting to set the spin center based on phantom part geometry.

When Navigation is set to Creo, set the spin center, press and hold the middle mouse button and then drag the pointer from one side of the figure window to the other for full 360° rotation.

Enhancement Details

Support for Exported Phantom Parts in Arbortext IsoDraw

Phantom parts in a figure saved as a 2D illustration file retain their phantom appearance in Arbortext IsoDraw.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	
Processes, Initiatives, and Best Practices	

Benefits and Description

Arbortext IsoDraw handles phantom parts as listed below:

- Draws thick and thin lines on phantom HLR parts with thick and thin gray Pens. Pen names are Phantom1 and Phantom2.
- Groups all phantom parts in the Phantom element group separate from the illustration structure.

Enhancement Details

Translate and Rotate Sections with Draggers

You can translate and rotate sections with draggers.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Select a section cut, and then click Home > Translation Mode or Home > Rotation Mode. The translation or rotation dragger appears on the section.
Processes, Initiatives, and Best Practices	

Benefits and Description

In Translation or Rotation mode, you can use the dragger to move or rotate the section cut plane the same way you use it to move or rotate parts. If you select a section cut plane and one or more parts, you can also use the dragger to move or rotate them together.

Enhancement Details

Unscrew Coupled Parts in Animations

There is an Unscrew dialog box.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	Unscrew dialog box
Processes, Initiatives, and Best Practices	

Benefits and Description

Use the Unscrew dialog box to apply the Unscrew animation effect to one or more sets of coupled parts and single parts. Coupled parts unscrew as a single unit along an axis relative to their bounding box. Specify a custom unscrew axis relative to a coordinate system, or use the unscrew axis Creo Illustrate selects automatically. Preview your Unscrew effect settings manually or automatically.

Enhancement Details

Validating an Illustration

You can validate an illustration by comparing the contents of an exported companion XML file to the sBOM, figures, and item lists in the .c3di file.

Product Information

Product	Creo Illustrate
PTC Support Release	Creo 2.0
Product Functional Area	Visualization
User Interface Location	In Structure Edit mode, click Home > Export, and then save the companion XML file. Or, publish the illustration and the companion XML file is saved along with the .pvz file.
Processes, Initiatives, and Best Practices	

Benefits and Description

The comparison can, for example, help you identify which parts are mapped to which rows in which figure's item list. You can also use it to verify item numbers in item list rows and figure callouts.

You can export a companion XML file for validation in Structure Edit mode by exporting and then saving the companion XML file. You can also publish the illustration and the companion XML file is saved along with the .pvz file.