



# What's New for Creo<sup>®</sup> View 4.0



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## What's New for Creo View MCAD

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## Batch Interference—Open an Interference in Creo View from Windchill

You can open specific interferences from Windchill batch interference results in Creo View, in addition to opening batch interference reports in Creo View.

Release: Creo View 4.0 with Windchill 11.0 M020

### Benefits and Description

You can open batch interference results in Creo View by selecting the interferences of interest. Interferences are selected when opened in Creo View. You do not need to open the report and then find and activate the interferences in the report listing. You may be viewing interferences that are not in the context of an interference report. For example, you may be viewing interferences associated to a Change Management Object such as a problem report. As long as the interferences are in the same report, you can open them directly in Creo View from the related table on the change objects property page from Windchill.

There is an interference report shortcut menu in Creo View from which you can reselect preselected interferences from Windchill, at any time.

## Setting the Default for Selection Filter

You can set the default selection filter.

User Interface Location: Click **File** ► **Options** and then click **Defaults**. In the left pane select **Model**, **Illustration Viewer**, **PCB Layout Viewer**, or **Schematic Viewer**. In the right pane, in the **Selection Method** box, select a selection filter.

Release: Creo View 4.0

### Benefits and Description

Previously the selection filter was set to **Smart**. In Creo View 4.0 you can set the default selection filter for different views. For example, if you usually work with models in Creo View, you can set the default **Selection Method** for models to **Parts**.

## Sorting Creo View Trees and Views

You can sort the attributed columns for the following trees and views in Creo View: Structure Tree, SBOM Tree, Model Tree, Interference Report View, Geometry Report View.

Release: Creo View 4.0



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## Benefits and Description

Column headers are displayed when the **Use tree columns** check box is selected in the **Creo View Options** dialog box **Global—Navigation** page. When **Use tree columns** is selected, even when only one column is displayed, you can use the column headers to sort the tree among siblings in the orders listed below:

- Alphabetic ascending order
- Alphabetic descending order
- Turn off sorting for that column

Each click in the column header cycles through the 3 methods of sorting. Preferences for sorting are applied to the Structure Tree following a structure edit, or after adding additional attribute columns. For dynamic structures from Windchill, a separate sorting state is saved. Therefore, it is possible to have the scenario described below:

- A default to not sort the Structure Tree of representations opened in Creo View. Consequently, the structure is sorted as it would be in the CAD system.
- A default to sort the Structure Tree of dynamic structures. Since they have no defined order, an alphabetic sort provides a logical display.

## Managing Creo View Search Queries

You can save Creo View search queries to local XML files.

User Interface Location: From the **Manage Queries** dialog box, click **Export**.

Release: Creo View 4.0

### Benefits and Description

In addition to loading search queries from XML files, you can also save search queries defined in to XML files which can then be loaded into Creo View. As a result, standard search queries can be managed external to the Creo View preferences, without having to manually create the XML query files. You can save search queries from the **Manage Queries** dialog box by selecting one or more search queries from the list and clicking **Export**. The selected search queries are saved to the specified XML file.

## Alternate 3D Transform Dragger

There is an alternate 3D dragger to move, rotate, or scale, selected parts, when the original dragger is out of view.

Release: Creo View 4.0

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### Benefits and Description

When zooming in close, the original dragger may not be visible or usable. When the original dragger is out of view, an alternate dragger is displayed. Use the alternate dragger to move, rotate, or scale selected parts. The arrow at the center of the alternate dragger points to the center of the original dragger.

## Importing STEP AP242 Files Is Supported

Creo View supports the opening and importing of STEP AP242 standard files.

Release: Creo View 4.0

### Benefits and Description

In Creo View you can open and import STEP AP242 format files using the updated client import filter. Display of STEP AP242 part-level annotations is also supported. These annotations appear in the **Model Annotations** tab as well as in the view. You can also show, hide, or isolate any annotation.

## Default Assembly-Level Color Is Stored in the PVS File

The default assembly-level color is stored in the PVS file.

Release: Creo View 4.0

### Benefits and Description

The Creo View 4.0 Adapter for Creo Parametric is enhanced to store the default color that is defined in a Creo Parametric assembly, in the Creo View PVS file. Creo View MCAD is enhanced to use the default color if it is defined in the .PVS file, instead of using the default color defined in the **Creo View Options** dialog box.

## Loading the Default View State in Creo View

You can load the default view state automatically.

User Interface Location: Click **File** ► **Options** and then next to **Showing** select **Global**. Under **General**, select **Loading** and then in the right pane select the **Autoload the PVS file's default view state** check box.

Release: Creo View 4.0

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## Benefits and Description

Select the **Autoload the PVS file's default view state** check box to load the default view state when the design, published from the Creo Parametric 4.0 adapter, is open in Creo View.



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## What's New for Creo View MCAD Adapters

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## MCAD Adapters: Additional Creo View Support

Release: Creo View Adapters 4.0

### Benefits and Description

Support is added for the products listed below:

- NX11
- Creo Elements/Direct Drafting 20.0
- Creo Elements/Direct Modeling 20.0
- Creo Parametric 4.0

## MCAD Adapters: Discontinued Creo View Support

Release: Creo View Adapters 4.0

### Benefits and Description

Support is discontinued for older CAD tools:

- CADD5 16.0
- SOLIDWORKS 2013
- NX 8
- Autodesk Inventor 2013

## Creo Parametric: Default Assembly–Level Color Is Stored in the PVS File

The default color for parts that do not have an explicit color assigned to their faces can be defined by the Creo View Adapter for Creo Parametric 4.0.

Release: Creo View Adapters 4.0

### Benefits and Description

The Creo View 4.0 Adapter for Creo Parametric is enhanced to store the default color that is defined in a Creo Parametric assembly, in the Creo View PVS file. Creo View MCAD is enhanced to use the default color if it is defined in the PVS file, instead of using the default color defined in the **Creo View Options** dialog box.

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## Creo Parametric: Combined State Publishing Filter Is Supported

The combined state publishing filter applied in Creo Parametric 4.0 is supported.

Release: Creo View Adapters 4.0

### Benefits and Description

The publishing filters applied in Creo Parametric 4.0 for combined states are supported when publishing to Creo View. Any combination states that are cleared in Creo Parametric are not displayed in Creo View.

## Creo Parametric: Default View State at Time of Publishing Is Supported

Publishing the default view state from Creo Parametric is supported. In addition, you can configure the default view state to be automatically loaded in Creo View.

Release: Creo View Adapters 4.0

### Benefits and Description

If the default view state is available in Creo Parametric, it is published to Creo View as the preferred view state and can be loaded automatically in Creo View.

## Creo Parametric: Appearance View State Is Supported

Appearances applied to combination states in Creo Parametric are supported during publishing.

Release: Creo View Adapters 4.0

### Benefits and Description

Appearances are introduced in Creo Parametric 4.0. Appearances applied to combination states in Creo Parametric are supported during the publishing process to Creo View. These appearances are applied when viewing the combination state in Creo View.

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## Creo Parametric: Font Support for Screen-Space Annotations Is Supported

Font support is available for screen-space annotations published from Creo Parametric 4.0.

Release: Creo View Adapters 4.0

### Benefits and Description

Screen-space annotations created in Creo Parametric 4.0 can be published as fonts and viewed in Creo View. The text fonts in Creo View support font styles including bold, italic, and underline text. In the Recipe Editor, the **Output text as lines** check box must be cleared for the fonts to be published. The check box is cleared by default.

## Creo Parametric: Publishing Annotations and Supplemental Geometry in Combination States Is Enhanced

Different supplemental geometries such as datums and annotations included in combination states in Creo Parametric, can be published to Creo View.

Release: Creo View Adapters 4.0

### Benefits and Description

Creo Parametric 4.0 supports the inclusion of supplemental geometry such as datum planes, points, curves, and so on in combination states as well as the ability to control the visibility in the combination states. With the Creo View 4.0 Adapter for Creo Parametric, this data can be published to Creo View. For supplemental geometry to be displayed in Creo View, the Creo View filters for datums must be selected in the filters dialog box.

## Creo Parametric: Additional Geometric Tolerance (GTOL) Symbols, Indicators, and Weld Symbols Are Supported

GTOL symbols and weld symbols are introduced in Creo Parametric 4.0 and are supported in Creo View.

Release: Creo View Adapters 4.0



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## **Benefits and Description**

Creo Parametric 4.0 introduces new GTOL symbols and indicators which can be published and viewed in Creo View. It is also possible to configure batch and interactive publishing for the symbol font options Legacy, ASME, and ISO.

New weld symbols introduced in Creo Parametric 4.0 can be published to Creo View as part of batch and interactive publishing.

## **Creo Parametric: Datum Target Annotation Element (AE) Is Supported**

Datum targets, when published from Creo Parametric to Creo View, are displayed with the correct icon.

Release: Creo View Adapters 4.0

## **Benefits and Description**

Datum targets are published to Creo View with the correct icon and are no longer created as Symbol Annotations. The new icon is supported only when publishing with the Creo View 4.0 Adapter for Creo Parametric 4.0.

## **Creo Parametric: Publishing of Non-Native 3D CAD Models Checked into Windchill Is Supported**

Publishing of non-native 3D CAD models from Creo Parametric to Creo View using the Creo View Adapter for Creo Parametric is supported.

Publishing of non-native 3D CAD models checked in by Creo Parametric 4.0 to Windchill, are converted to Creo View using the Creo View Adapter for Creo Parametric.

Release: Creo View Adapters 4.0

## **Benefits and Description**

In Creo Parametric 4.0, you can check in 3D parts and assemblies from NX, CATIA V5, SOLIDWORKS and Creo Elements/Direct Modeling into Windchill, without requiring these CAD tools. The Creo View Adapter for Creo Parametric and the Windchill Visualization Services are also enhanced to enable the publishing of these non-native CAD models in Windchill.

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## Inventor: Publishing of Autodesk Inventor Presentations

You can publish Autodesk Inventor presentation files to Creo View format.

Release: Creo View Adapters 4.0

### Benefits and Description.

The Creo View Adapter for Autodesk Inventor supports the publishing of Inventor presentation `.ipn` files. When loaded in Creo View, the active scene from the Inventor presentation is visible and exploded view states similar to what is seen in Inventor, are displayed.

## JT: Publishing and Display of Layer Groups Is Supported

Layer Groups for JT files can be converted. You can use the layer groups in Creo View to control the display of geometry.

Release: Creo View Adapters 4.0, Creo View 4.0. If used with Windchill, Windchill M020.

### Benefits and Description

JT files can contain layer groups that define the off or on display state of layers in the data set. Layer states can be converted to Creo View and the resulting layer group information stored in the `.pvs` file. In the Creo View client, these layer groups and their layers, control the geometry display. This requires an extension to the `.pvs` file format if used with Windchill, therefore, Windchill 11.0 M020 is required.

## JT: Ultra Lightweight Precise JT Format (ULP) Is Supported

The ULP version of JT is added to the JT Adapter.

Release: Creo View Adapters 4.0

### Benefits and Description

Support is provided for the ULP version of the JT file format. To have this support requires that the Creo View JT Adapter is installed on the local machine.

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## What's New for Creo View ECAD Adapters?

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## **ECAD Adapters: Additional Creo View ECAD Support**

Release: Creo View Adapters 4.0

### **Benefits and Description**

Support is added for the products listed below:

- Cadence 17.2
- Mentor Graphics VX 1.2 and VX.2
- Zuken CR5000 18.0
- Zuken CR8000 2016.000–1
- Altium Designer 16.0 and 16.1

## **ECAD Adapters: Discontinued Creo View ECAD Support**

Release: Creo View Adapters 4.0

### **Benefits and Description**

Support is discontinued for older CAD tools:

- Cadence 16.5
- Mentor Graphics Design Capture—All versions
- Zuken CR5000 14.0, 15.2
- Altium Designer 10, 12

## **Update to Zuken CR-5000 Board Designer Adapter**

Release: Creo View Adapters 4.0

### **Benefits and Description**

The Zuken CR-5000 Board Designer PCB Layout adapter uses the actual pin name which is in synchronization with the schematic design. Previously the pin number was used, which was an internally generated number.