PTC®

Creo™ View 1.0 Help Topic Collection

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

Contents

1

Creo View Overview

About Creo View

Creo View enables you to view and interrogate ECAD designs and 3D MCAD products and models, assemblies, drawings, images, and documents. You can collaborate on products of different information types without the native applications. Using Creo View, you can do the following tasks:

- View a structure
- Edit the appearance of a structure by changing part color, render mode, location, and orientation
- Mark up a structure with annotations, measurements, and construction geometry
- Group parts
- Create cross sections
- Move parts and change their appearance, such as creating exploded views and cross sections

With Creo View, team members can view, mark up, interact, and collaborate on all forms of digital product data, and visual information is shared across the enterprise.

About Creo View Editions

Creo View comes in two licensed editions and one unlicensed edition:

- Creo View—Includes all functionality, is either MCAD or ECAD based, and some applications require additional licenses.
- Creo View Lite—Provides limited viewing and markup functionality and interoperability with Windchill.
- Creo View Express—Provides basic viewing functionality without a license.

The Creo View license is configured for MCAD or ECAD. The Lite edition of the other configuration is provided. You can open the Creo View edition of the other configuration if you need to work in it.

Depending on the Creo View license configurations set up by the systems administrator, you can retrieve licenses for your workstation according to your tasks. This flexibility enables efficient license sharing.

The following modules require additional licensing:

- Animator
- Interference Analysis
- Design Check
- Massive Assemblies

The license for these modules can be set to be retrieved at startup or when required.

To Set the Version and License Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global** ▸ **Startup**. The **General-Startup** dialog box opens.
3. Clear **Choose Edition at startup** to open Creo View according to these settings at startup.
4. To determine the Creo View edition at the next startup select one of the **Choose Creo View Edition** options:
   - **Creo View Lite** .
   - **Creo View MCAD**.
   - **Creo View ECAD**.
   - **Creo View MCAD and ECAD**.
5. Under **MCAD License Options** ▸ **Edition** and **ECAD License Options** ▸ **Edition**, if you did not select **Creo View Lite** , select one of these options:
   - **Load at startup**
   - **Load when required**
6. Select a license loading option for each of theses applications:
   - **Design Check**
   - **Interference Analysis**
   - **Animator**
   - **Massive Assemblies**
7. To update your license, under **License Information** type the **License Server** name in the box.
8. Click **Apply** to apply the options. Click **OK** to apply the options and close the dialog box.
9. Restart Creo View to open with the new settings.

2

Using Creo View Help

About the Context Sensitive Help

Creo View Help is context sensitive. To access a relevant topic, hover the pointer over a button or a dialog box, and then press F1.

To access the Help system, click  in the upper right corner of the Ribbon or press F1.

To access a help topic about an open dialog box, click  in the upper right corner of the dialog box.

3

Navigating the User Interface

About the Creo View User Interface

The Creo View user interface is divided into these elements:

- Viewing
  - Graphics area—Shows a visualization of the file. You can change the view to display 3D models, 2D drawings, images, and documents. Use the commands on the shortcut menus to control the view and to set viewing options. Several views (drawings, images and documents) can be open at the same time. Click to toggle between the open views.
- Navigating the structure
  - Primary panel
  - Upper data panel

  The Panels show information about the product structure. You can click the arrow along the panel's sides or drag the edges of the panel to show or hide or to resize it. You can also access a panel command in the status bar in the bottom right corner of the Creo View window.
- Performing commands
  - Ribbon
  - Quick Access Toolbar
  - Dialog boxes
- Configuring
  - Creo View Options
- Getting information
  - Lower data panel
  - Status bar—Shows the name of the current view and selection and unit settings.

About KeyTips and Keyboard Shortcuts

You can work more efficiently by using KeyTips and keyboard shortcuts to navigate the user interface and activate commands. By typing a keystroke combination, you can activate most of the ribbon commands. KeyTips indicate which keys you should press to activate a command. The KeyTips for the ribbon tabs appear when you press the ALT key. You can press the letter that is indicated in the KeyTip .

To Use KeyTips

1. Press the ALT key. The KeyTips appear on the ribbon.
2. Press the letter or letters in the KeyTip over the tab that you want to open. The tab opens.
3. To hide the KeyTips, press the ALT key.

Keyboard Shortcuts

Command	Keyboard Shortcut
Clear Annotations	CTRL + SHIFT + DEL
Clear Measurements	CTRL + DEL
Close Window	CTRL + F4
Context Help	F1
Delete	DEL
Exit current mode (return to select mode)	ESC
Find	CTRL + F
Help Topics	F1
ONLY display View window	CTRL + F8
Print	CTRL + P
Render Shaded	F9
Render Shaded with Edges	F10
Render Hidden Line Removal	F11
Render Wireframe	F12
Rotate mode	F4
Select All	CTRL + A
Spin Center	CTRL + . (period)
Translate mode	F2
Zoom All	CTRL + SHIFT + A
Zoom Out	CTRL + -
Zoom In	CTRL + +
Zoom Selected	CTRL + SHIFT + S
Zoom Window	CTRL + SHIFT + Z
Start or stop animation playback	SPACEBAR

Undo and Redo Commands

Creo View provides commands to undo and redo actions in the view, such as moving parts, editing properties, sectioning changes, annotations, and creating construction geometry. **Undo** and **Redo** do not apply to standard view manipulation such as panning or rotation, or to print, save, copy, or option commands.

- To undo or redo only the most recent action, left-click the **Undo** (Ctrl+Z) or **Redo** (Ctrl+Y) button on the quick access toolbar.
- To display a list of multiple recent actions for selection, click the arrow next to the **Undo** or **Redo** command icon. When you make a selection from the list, the undo/redo action applies to all commands in the list that were executed prior to that selection in the list. In other words, you can select the most recent command, or a range of commands to undo/redo.
- You can use the **Limit undo/redo steps to** options on the **General** option page to control the number of actions that are available to undo or redo.

Note

A saved annotation set cannot be undone.

Working with Panels

About Panels

Panels are portions of the Creo View window that display information about the structure or design. The panels are below and to the left of the graphics area. Click the arrows to show and hide panels.

Panel Name	Position	Information Displayed
Primary panel	Left side of the Creo View window	Structure, viewables, annotations, ECAD substructures
Upper data panel	To the right of the Primary panel	Layers, groups, search results, view content, ECAD color and visibility, Design Check and Interference Detection results
Lower data panel	At the bottom of the Creo View window	Attributes, messages, Bill of Materials, Animation Keyframe Editor

To Customize the Display of Panels

1. On the Status bar, toggle the display of one or more panels:
   -  —Primary panel
   -  —Upper data panel
   -  —Lower data panel
2. To hide everything but the graphics area, click  .

About the Structure Tree

In Creo View, use the Structure Tree to display a hierarchical view of all components in the structure. The root structure is at the top of the Structure Tree, and its subordinate features and parts are below it. Use this view to show all the parts of the model, and to load or to unload parts. Items are represented in the Structure Tree with these glyphs:

-  —Top-level structure
-  —Subassembly
-  —Part

Large structure trees can be cumbersome to work with, and the node names displayed can be meaningless to the Creo View user. To improve performance and make it easier to relate to the Structure Tree display, you can do the following:

- Filter some of the nodes out of the view.
- Change the node names displayed.

When you click the Structure Tree, the **Structure** tab on the Ribbon is activated.

About Messages

The  Messages pane in the lower data panel lists informational, debug, warning, and error messages logged during your session. You can configure the information that is displayed with a message and the maximum file size for a message.

To Log Messages

1. Make sure that the  pane of the lower data panel is open.
2. Right-click and choose **Capturing Messages** from the shortcut menu. Message logging is enabled.

3. To set the types of messages logged, right-click and choose **Capture Types**, and then select one or more message types.
4. To display information:
    a. Use the context menu or options to turn message logging on. You can also enable message logging and specify log file size using the general options. When message logging is on, an icon appears in the status area beside the units of measure.
    b. Messages appear in the Messages pane in the lower data panel. The type of message is indicated by a corresponding icon.
    c. In the Messages pane, use the context menu to control the types of messages that are logged. You can click a column header to sort by that column. The context menu lets you select which columns to display (Time, Thread, Source, Message). You can also select **Show Detail** to display the Message Detail window, which shows the message type, thread, source, and text. In this window, you can choose to **Copy All** or **Copy Only Message** to paste into another application.
    d. In the **Messages** context tab that appears in the ribbon area, you can select to Show Detail, Copy, specify visible columns, display message options, and clear messages. You can also select which types of messages to capture: Info, Debug, Warning, and Error.

About the Views Gallery

The Views gallery appears in the **Views** group on the **Home** tab. Use it to manage views, to select a recently opened view, to close the current view (CTRL + F4), to close all views, or to show multiple views.

Recently opened views appear as thumbnails. Click a thumbnail to open the view.

About the Status Bar

The status bar shows the loaded view, available selection options, and other relevant information depending on the active file:

- Click a window arrangement icon to change the placement of windows in the Creo View interface
- The progress bar, shown in all views, indicates the percentage of completion of the task being executed, which advances from 0 to 100 as the task is completed.

- The status bar also shows the number of selected items in the view.
- Use the selection menu to choose the type of selection (smart, parts, faces, or edges), as well as units of measure.

4

Customizing Creo View

Customizing Tooltips

About Customizing Tooltips

When you place the pointer over a part in a structure, a tooltip appears in the Status Bar. By default, it contains the part name. You can customize the tooltip to include other attributes, values, or any other text. You can select from these categories:

- System Variables
- PVS File Properties
- PVZ File Properties
- EDA File Properties
- Creo Parametric Parameters
- Creo Parametric Component Parameters

To Customize Tooltips

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **Model**, **PCB Layout Viewer**, or **Schematic Viewer**, click **Tooltips**. The appropriate **Tooltips** options open.
3. Set one or more of the following options:
   - **Show Part Tooltips**—Displays the tooltips for parts.
   - **Use Default Tooltip (Part Name)**—Displays the default tooltip for the MCAD model part name. If a custom tooltip is defined in the **Tooltip Editor**, this text is shown in the tooltip.
   - **Show Tooltips**—Displays the ECAD tooltips.
   - **Exclude Overlays**—Displays tooltips of objects underneath overlays, but not tooltips of the overlays.
   - **Exclude Planes and Conductive Figures from Pre-selection of Nets**—Prevents large objects from being highlighted when you move the pointer over them. This does not affect the selection highlighting.
4. In the **Tooltip Editor**, type additional information for the tooltip.
5. To include attribute names and variables in tooltips, perform the following operations:
   a. In the **Tooltip Editor**, right click and choose **Insert Variable** from the shortcut menu. The **Insert Variable** dialog box opens.
   b. Select variables in the **Category** and **Attribute** boxes.
   c. Click **Insert Name** or **Insert Value**. The variable is inserted.

d. Repeat steps b-c for each variable to insert.

e. Click **Close**.

6. In the **Creo View Options** dialog box, click **OK**.

Example: Custom Tooltip Syntax

Any text surrounded by < > is replaced with the appropriate attribute name or value. For example, the text `<System Variables.Part name.Name>` is replaced by the name of the component path attribute in the system variables group. Similarly, the text `<System Variables.Part name.Value>` is replaced by the value of the same attribute.

Customizing the Quick Access Toolbar

About the Quick Access Toolbar

The Quick Access Toolbar is located at the top of the Creo View window. By default, it contains the following commonly used commands:

-  —Delete
-  —Redo
-  —Save All
-  —Undo
-  —Customize

You make the following changes to customize the toolbar:

- Move the toolbar
- Add commands
- Remove commands

To Add a Command to the Quick Access Toolbar

1. On the Quick Access Toolbar, click  . The **Customize Quick Access Toolbar** options open.
2. Select one or more of the following commands to add it to the toolbar:
   - **Open**
   - **Save All**
   - **Print**

- **Undo**
- **Redo**
- **Delete**
- **Search**
- **Select Mode**

3. To add a command from the Ribbon or from a dialog box, select the command, right-click, and choose **Add to Quick Access Toolbar** from the shortcut menu. The command is added.

To Remove a Command from the Quick Access Toolbar

1. On the Quick Access Toolbar, right-click and choose **More Commands**. The **Customize-Main** dialog box opens.
2. Select a command from the **Customize Quick Access Toolbar** list and click **Remove**. The command is removed.
3. Click **Apply** to apply the settings and keep the dialog box open, or click **OK** to apply the settings and close the dialog box.

To Move the Quick Access Toolbar

1. On the Quick Access Toolbar, click  . The **Customize Quick Access Toolbar** menu opens.
2. Select one of the following commands:
   - **Place Quick Access Toolbar below the Ribbon**
   - **Place Quick Access Toolbar above the Ribbon**

   The toolbar is moved.

To Set Quick Access Toolbar Options

1. On the Quick Access Toolbar, click  . The **Customize Quick Access Toolbar** menu opens.
2. Click **More Commands**. The **Creo View Options** dialog box opens and the **Customize-Main** options open on the right.
3. In the **Choose commands from** box, select a list of commands. The command list is displayed in the box on the left.
4. Select a command and click **Add**. The command is added to the **Customize Quick Access Toolbar** box on the right.

5. Repeat step 4 until you have added all the desired commands to the **Customize Quick Access Toolbar** box.
6. To move the Quick Access Toolbar in the Creo View window, select or clear the **Place Quick Access Toolbar below the Ribbon** check box.
7. Click **Apply** to apply the changes and continue setting options, or click **OK** to apply the changes and close the dialog box.

Setting Options

About Creo View Options

Options in the **Creo View Options** dialog box are applied as follows:

- **Global**—Applies to all views of a structure.
- **Active View**—Affects the active view only.
- **Defaults**—Applies the default values to future views.

You can set global and default options before loading data.

The system administrator can lock view options for a workstation. When a view option is locked,  appears alongside it.

To Set General-Main Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens on the right.
2. In the **Showing** box, select **Global**, and then on the left select **General**. The **General-Main** page opens.
3. Select one or more of these check boxes:
   - **Display Thumbnails in View Gallery**—Clear this box if generating thumbnails is time-consuming.
   - **Show dialog to confirm closing of all active views**
   - **Switch to perspective view mode automatically when Explore navigation method is selected**
   - **Show alert message for advanced search with duplicate colors**
4. Set the **Limit undo/redo steps to** box.
5. In the **Limit the minimum page size of paged lists to** box, type or select the minimum page size of paged lists.

6. In the **Limit the minimum page size of the attribute list to** box, type or select the minimum page size of the attribute list.
7. Set or clear the **Show alert message for advanced search with duplicate colors** check box.
8. Set or clear **Log messages** with its file size option.
9. Set **Part highlighting mode** to **Wireframe** or **Filled**.
10. Click **Apply** to apply the options. Click **OK** to apply the options and close the dialog box.

General Creo View Options

Select general options from the **General-Main** Options dialog box.

Option	Description
Display Thumbnails in View Gallery	Displays open views in the Views display grid.
Show dialog to confirm closing of all active views	Displays a warning message before closing unsaved data.
Switch to perspective view mode automatically when Explore navigation method is selected	Activates a perspective view automatically in Explore navigation mode.
Show alert message for advanced search with duplicate colors	Displays a warning message if you select the same color for more than one query when performing a color-coded search.

About Locking Option Settings

The Creo View administrator can lock option settings and commands as follows:

- Any individual option or the entire option page can be locked. When a option setting is locked, a lock icon appears alongside the setting on the Preference page.
- Any command can be locked. A tooltip indicating that the command has been disabled appears when you hover the cursor over a disabled command.
- If you have set `PVIEW_DISPLAY_COMMANDS=1` config options, the Tooltips also show the command ID.

Note

You cannot disable interface elements such as tabs and groups, but you can disable an individual command, such as ***Print****.*

To Lock Options

There are three ways to lock option settings and disable commands:

- Download from server
- Update `{install_dir}/Options/ProductView/admin_prefs.xml` file.
- Update `%APPDATA%/ptc/ProductView/user_prefs.xml`

There is a `readme.txt` file in the directory `{install_dir}//preferences/ProductView` that provides information about disabled commands and locked option settings.

To Reset Options to the Defaults

1. In the **Creo View Options** dialog box, click one of the following buttons:
   - To reset a specific group of options, with that group open, on the right, click **Reset to Default**. The default settings are restored.
   - To reset all options, on the left, click **Reset All to Default**. All default settings are restored.
2. Click **OK** to apply the settings and close the dialog box, or click **Apply** to apply the settings and continue setting options.

Customizing the Palette

About Customizing the Palette

The Creo View palette includes 16 basic colors. In Creo View you can add any number of custom colors. Set the palette to display all colors or only custom colors. Use the **Creo View Options** menu to create, save, and edit new colors or to delete them. You can also add a name to each color that appears in a tool tip. In Creo View Lite, you can view custom colors, but you cannot modify the palette.

To Add Custom Colors to the Palette

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**. On the right, the global options open.
3. On the left, under **General**, select **Color Palette**. On the right, the **General-Color Palette** options open.
4. Click **Add**. The **Choose Color** dialog box opens.

5. Select the values for red, green, and blue in one of these ways:
   - Under **Settings**, drag the sliders next to **Red**, **Green**, and **Blue**, or type values in the boxes.
   - Select a color on the color wheel, and then drag the slider to adjust the tint.
6. Click **OK**.
7. Repeat steps 5–7 for each custom color.
8. To add a tooltip, select a color and click **Color Tooltip**. The **Name Color** dialog box opens.
9. Type a name in the **Text** box, and then click **OK**.
10. To exclude the basic colors from the palette, in the **Creo View Options** dialog box, select the **Use Only Custom Colors** check box.
11. Click **OK**.

To Edit or Delete Custom Colors

1. In the **Creo View Options** dialog box, under the **General-Color Palette** options, select a color to edit and click **Edit Color**.
2. Adjust the **Red**, **Green**, and **Blue** values, and then click **OK**. The color is edited.
3. To delete a custom color, select it and click **Remove**. The color is deleted.

5

Opening Structures and Designs

About Opening Structures and Designs

You can open files that contain MCAD data, ECAD designs, or both. The **File** menu contains a list of recently opened files. Open Creo View files, or import one of the supported external formats.

When you open a file, it is not automatically loaded. Instead, you can select one or more viewables in the file to load. In MCAD structures, within the model view, you can select one or more parts or subassemblies to load, or you can load the entire structure. A Creo View structure contains information about the makeup, organization, and appearance of the parts of a product. In Creo View, you can edit the structure of a product. A structure file has one or more of the following MCAD viewables:

- Model
- Image
- Drawing
- Document
- Illustration

A structure can also have one or more of the following ECAD viewables:

- Schematic
- PCB Layout
- Overlay

After you open the structure, you can open one or more viewables associated with it. In addition to viewable files, the following information can be linked to the structure file:

- Appearance of parts
- Markups
- Attributes
- Groups of parts
- Sections

To Open a Structure or Design

1. Click **Open**. The **Open File** dialog box opens.
2. Select the folder that contains the structure or design file to open, or type a name in the **Search** box.

3. To filter the files by type, select a file type from the **Type** box.
4. Select the file, and then click **Open**.

About Creo View File Formats

Creo View files use the following suffixes:

Type	Extension

Structure	`*.pvs`
	`*.ed`
	`*.edz`

3D model	`*.ol`
Annotation	`*.pva`
Interference report	`*.pvc`
PCB Layout Schematic	`*.eda`
Compressed package with one or more of the types above	`*.pvz`

Loading Data Automatically

About Automatic Loading of Data

Loading options control how a structure opens in Creo View. You can set an automatic file load for all files, for a single file, or for no files. You can also set an automatic load for branch links.

To Load Data Automatically

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** window opens.
2. On the left, under **General**, click **Loading** . The **General-Loading** options open on the right.
3. Select an option from the **Load Creo View data as** box:
   - **Autoload single file**—If there is a single node in the structure with 3D geometry attached, this option loads only that part. If there are no parts with 3D geometry, Creo View loads one of the other files types, using the first one in the hierarchy (drawing, document, and image, respectively). If there are multiple parts with 3D geometry, nothing is loaded. This is the

default setting, and is commonly used when you want to view only one part or file.

- **Autoload all files**—Loads the entire product structure.
- **No autoload**—Disables automatic loading. This option is generally selected when viewing a large assembly.

4. Select or clear the **Autoload if Structure has 3D Thumbnails** box.
5. To enable automatic loading and expanding of branch links, select the **Autoload branch links** box.
6. Click **Apply** or **OK**.

To Set Load Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the left pane, under **General**, select **Loading**. The **General-Loading** options open in the right pane.
3. To set the way in which files are loaded when Creo View is opened, select one of the following options in the **Load Creo View data as** box:
   - **Autoload single file**—One file is loaded.
   - **Autoload all files**—All the files are loaded.
   - **No autoload**—No files are loaded automatically.
4. To automatically load structure files that include 3D thumbnail previews, select the **Autoload if Structure has 3D Thumbnails** check box.
5. To automatically load a structure together with its branch links, select the **Autoload branch links** check box.
6. Click **Apply** to apply the settings and continue setting options, or click **OK** to apply the settings and close the dialog box.

6

Creo View Interface

About Importing and Exporting Files

You can import external MCAD or ECAD files, add annotations, and save the file as a Creo View file. By default, the import filters for most external formats are automatically installed when you install Creo View. However, the JT filter must be manually installed. You can customize the installation to include only the filters you need. In Creo View, you can export Creo View MCAD structures to 3D files, and you can export MCAD and ECAD views to these types of 2D files:

- Vector
- Image

About Saving External File Formats in Creo View

You can save external files that you open as Creo View files.

In Creo View Lite and Creo View, you can save data to the following formats:

- PVZ—Includes files for all viewables and annotations
- PVS—Saves the information as a directory and references the viewable and annotations files

The following ECAD file types are not saved as EDA files when you save the structure:

- Gerber
- ODB++
- IDF

Instead, the resulting PVZ or PVS file includes or references the native file for these viewables.

In Creo View, you can save MCAD files as image files, 2D vector files, or 3D files, and ECAD files as image files or EDA files.

To Save an External File Format in Creo View

1. To overwrite the current structure, click **File ▸ Save All**. The files are saved.
2. To create a new structure, click **File ▸ Save As**, and then click one of these options:

- **Save All As PVS**
- **Save All As PVZ**
- **Save Copy Of All As PVS**
- **Save Copy Of All As PVZ**

The **Save As** dialog box opens. Proceed to Step 5.

3. To save an ECAD view as an EDA file, when working in Creo View, click **Save Current View As** ▸  **As an EDA file**.
4. The **Save View as EDA** dialog box opens.
5. Select a location for the file.
6. In the **File name** box, accept the default file name, or type a new one.
7. Click **Save**.

Importing MCAD and ECAD Files

Supported MCAD File Formats

You can open an IGES or MicroStation file as either a 2D or a 3D representation. You can open the following file types in Creo View:

File	Type	Extension
IGES	2D drawing 3D model	`*.igs` `*.iges`
STEP	3D model	`*.stp` `*.step`
JT	Parts and assemblies only	`*.jt`
VRML	3D model	`*.wrl` `*.vrml`
Stereolithography	3D model	`*.stl` `*.sla`
MicroStation	2D drawing 3D model	`*.dgn`
PTC Visualizer	3D model	`*.gbf` `*.gaf`
Drawing	2D drawing	`*.hp2` `*.hpg`

File	Type	Extension
		*.pgl *.plt *.hgl *.dxf *.dwg *.dwf *.dwfx
	3D drawing	*.dwf *.dwfx

Supported ECAD File Formats

You can open these ECAD file types as data in Creo View:

File	Type	Extension
IDF	ECAD-MCAD Collaboration	*.emn *.emp
IFF	Schematic	*.iff
DSN	Design Entry CIS—Schematic SPECCTRA—PCB Layout	*.dsn *.cct
CADIF	Schematic	*.paf *.cdf
Cadence Allegro Package Designer	PCB Layout	*.mcm
GenCAD		*.cad
Cadence Allegro PCB		*.brd
EDIF	Schematic	*.edf *.eds *.edi
Mentor Graphics Board Station Neutral		neutral.mgn tech.mgn traces.mgn

File	Type	Extension
		`design_ascii`
Mentor Design Capture DCE 7.9.1		`*.pjr`
Dx Designer		`*.pjr` `*.proj`
Zuken ASCII		`*.pcf` `*.can`
Mentor PADS ASCII	PCB Layout	`*.asc`
Mentor Expedition Requires installation of Mentor_EE2007_Interface		`*.hkp`

You can open these file types as data or as an overlay:

File	Extension

Gerber	`*.art`
	`*.grb`
	`*.pho`

ODB++	`*.tar`
	`*.Z`
	`*.tgz`
	`*.gz`
	Matrix file

Opening Creo Parametric Files in Creo View

You can open Creo Parametric files in Creo View. The opened files have the following characteristics:

File	Type	Comment
Part	`*.prt`	Bulk parts do not have geometry, however, they appear as regular parts.
Accelerator for a part family table	`*.xpr`	
Assembly	`*.asm` `*.mfg`	

File	Type	Comment
Accelerator for an assembly family table	`*.xas`	
User-defined feature	`*.gph`	
Drawing	`*.drw`	
Format	`*.frm`	These files behave like drawings.
Layout	`*.lay`	
Diagram	`*.dgm`	
Report	`*.rep`	
Section	`*.sec`	
Picture	`*.pic`	

Note

You can also open drawings that include shaded views.

You can also open file formats that have been converted by Creo Parametric:

File	Type	Comment
Drawing	`*.dwg` `*.dxf`	When a Creo Parametric drawing is converted to these file types, the colors are mapped to AutoCAD colors. In some instances the result may not match the original color.

Saving Display Information in Creo Parametric

When you save a Creo Parametric file with the intent of opening it in Creo View, make sure that you set the following configuration options:

- `save_model_display`—Select one of the following values:
  - `shaded_lod`
  - `shaded_low`
  - `shaded_high`
- `save_drawing_picture_file`—`both`
- `sketcher_save_preview_image`—`yes`

Exporting MCAD and ECAD Files

About Saving Views to External File Formats

You can save any view as an image file. In Creo View, you can save a drawing or an ECAD view as a 2D vector file, and you can save a model view as a 3D file. When you save to a 3D format, the result is a file that contains the parts selected in the graphics area. If nothing is selected, all the loaded and visible parts in the current view are saved.

Saving Views to External File Formats

You can export a view to DXF or to a 3D format only in Creo View. You can save a view in these formats:

View Type	File Type	Format	Extension
MCAD and ECAD	Image	BMP	`*.bmp`
		GIF	`*.gif`
		JPEG	`*.jpg`
		PNG	`*.png`
		TARGA	`*.tga`
		TIFF	`*.tif`
			`*.tiff`
MCAD	2D Vector	CGM	`*.cgm`
		DXF	`*.dxf`
		CGM	`*.cgm`
MCAD	3D	IGES	`*.igs`
		STEP	`*.stp`
		VRML	`*.wrl`

To Save a View as an Image

1. Click **File** ▸ **Save Current View As** ▸ **As an image file**. The **Save View As Image** dialog box opens.
2. In the **File name** box, type a name or accept the default.
3. In the **Type** box, select a file format.

4. Click **Save**. The **Save View As Image- Settings** dialog box opens.
5. Under **Units**, select the unit of measurement for the image:
   - **Inches**
   - **Centimeters**
   - **Pixels**
6. Under **Width** and **Height**, set the dimensions for the image.

   Note

   *When the **Lock aspect ratio** check box is selected, the proportions of the image are preserved. You can change one dimension and the other is updated automatically.*
7. To set a resolution for images measured in inches or centimeters, in the **Resolution** box, type a number of pixels per unit.
8. Click **OK**. The settings are applied and the image is saved.

To Save a View as a DXF File

1. Click **File** ▸ **Save Current View As** ▸ **As a 2D vector file**. The **Save View As Vector** dialog box opens.
2. In the **File name** box, type a name or accept the default.
3. In the **Type** box, select **DXF**.
4. Click **Save**. The **Save View As Vector**–**Settings** dialog box opens.
5. Under the **Zoom Setting**, select an option:
   - **Current Zoom**
   - **Zoom All**
6. Click **OK**. The view is saved.

To Save a View as a CGM File

1. Click **File** ▸ **Save Current View As** ▸ **As a 2D vector file**. The **Save View As Vector** dialog box opens.
2. In the **File name** box, type a name or accept the default.
3. In the **Type** box, select **CGM**.
4. Click **Save**. The **Save View As Vector** dialog box opens.
5. Under **Zoom Setting**, select a zoom option for the view:
   - **Zoom All**
   - **Current Zoom**

6. To create a file that meets ATA graphics standards, under **Format**, select the **Save as CGM ATA Profile 5** check box.
7. Click **Save**. The settings are applied and the file is saved.

Saving a View as a 2D Vector File

When you save a view as a 2D vector file, you can set file properties according to the format you select:

- CGM:
  - Unit of measurement
  - Dimensions of the image
  - Resolution
  - Compliance with ATA standards
- DXF:
  - Number of sheets

    Note

    DXF files cannot contain multiple sheets. When you save to DXF all the sheets of a multi-sheet drawing, one file is created for each sheet. The sheet number `n` *is added to the file name according to the syntax* `filename_n.dxf`.
  - Zoom setting for a single sheet

To Save a View as a 3D File

1. Click **File** ▸ **Save Current View As** ▸ **As a 3D file**. The **Save View As 3D** dialog box opens.
2. In the **File name** box, type a name or accept the default.
3. In the **Type** box, select a file format:
   - **IGES**
   - **STEP**
   - **VRML**
4. Click **Save**. For large files, the **Save View As 3D** dialog box opens with a progress bar.

Data Saved to the VRML File

When you save a view as a VRML file, the result of the operation is one, compressed VRML 2.0 file. VRML views include lights within a scene, while Creo View views do not. When you save to this file format, a generic ambient light is created for the VRML view. In addition, the following data is included:

Item	Property
Structure	Part names Hierarchy
Part	Tessellation Color Opacity Visible layers
View	Perspective Field of View setting Camera position and orientation

Data Saved to the STEP or IGES File

IGES does not support hierarchical structures with multiple parts. When you save a view as an IGES file, all of the structure's data is saved to a single part file. When you save a view as a STEP or IGES file, the following data are included:

Item	Property
Structure	Flat list without part names
Part	Color BREP as quilts only

7

Viewing the Structure

Displaying Views

About Views

You can display up to 12 views at once in the graphics area. This includes viewables of different MCAD and ECAD types and saved view states. Thumbnails of the open views appear on the **Home** tab of the ribbon in the **Views** group. Reducing the number of views visible in the graphics area hides the extra loaded views, but it does not close them or remove their thumbnails from the **Views** group.

Note

A view does not open automatically when you open a file. After you open a file, you can select one or more views to load.

When one or more views are open in the graphics area, additional views open as follows:

Active View	Action	Result
The same type of viewable as the new view to open	Double-click. Drag the view into the graphics area.	Replaces the active view with the new view.
A different type of viewable than the new view to open		Opens a new view.
Any	Right-click and choose **Open in New View**.	

When you open an ECAD view, the design's  substructure pane opens in the primary panel.

To Manage the Display of Views

1. On the **Home** tab, in the **Views** group, click  . The views display grid opens.
2. To manage the number of views displayed, in the views display grid, do one of the following operations:
   - Multiple views—Move the pointer over the number and arrangement of views that you want to display, and then select a square for the lower right view. The graphics area is divided.
   - One view—Select the top left square. One view is displayed in the graphics area.
3. To close one or more views, do one of the following operations:

- Current view—Press CTRL+F4 or, on the **Home** tab, in the **Views** group, click  .
- All views—On the **Home** tab, in the **Views** group, click  .

To Open an MCAD or ECAD View

1. Click  . The Viewables pane of the primary panel opens.
2. Select the partition for the type of viewable to open.
3. Do one of the following operations:
   - Double-click the viewable or a view state
   - Drag the viewable or view state into the graphics window
   - Right-click the viewable or a view state and choose **Open in New View** from the shortcut menu.

To Show Multiple Views

1. On the **Home** tab, in the **Views** group, click  .
2. Move the pointer over the number and arrangement of views that you want to display, and then select a square for the lower right view. The graphics area is divided.

To Revert to One View

1. On the **Home** tab, in the **Views** group, click  .
2. Select the top left square. One view is displayed in the graphics area.

Navigating and Selecting

Using the Mouse

About the Mouse Navigation Modes

In Creo View, you can set the mouse navigation mode to one of the following:

- **ProductView**
- **Creo**

Select the scheme with which you are most comfortable. The default type of navigation for both modes is Inspection. This method works with both orthographic and perspective viewing. In the **Creo View Options** dialog box, you can also select **Explore** to use an advanced fly mode for 3D navigation. In **Explore** mode, you can view an object from different depths and angles. This method works only with perspective viewing. The **Explore** setting maximizes the effects of fly navigation.

To Set the Mouse Navigation Mode

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **General**, click **Navigation**. The **General-Navigation** options open on the right.
3. In the **Navigation model** box, select a mode:
   - **ProductView**
   - **Creo**
4. To apply the **Explore** mode for navigation in the Model View, follow these steps:
   a. In the **Showing** box, select **Defaults**.
   b. On the left, click **Model**. The **Model-Main** options open on the right.
   c. Under **3D Navigation Method**, select **Explore**.
5. Click **OK** to apply the changes and close the dialog box, or click **Apply** to apply the changes and continue setting options.

To Set Navigation Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **General**, select **Navigation**. The **General-Navigation** options open on the right.
3. To set the options for highlighting of selected parts, surfaces, edges, and points:
   - Select or clear the **Select using color** check box.
   - To change the highlighting color, click the color box and set a new color in the **Choose Color** dialog box.
4. To set the options for secondary selection color of parts, surfaces, edges, and points:
   a. Select or clear the **Secondary selection color** check box.
   b. To change the highlighting color, click the color box and set a new color in the **Choose Color** dialog box.

5. To set the options for pre-selection highlighting of parts, surfaces, edges, and points:
   a. Select or clear the **Enable pre-selection highlighting** check box.
   b. To change the highlighting color, click the color box and set a new color in the **Choose Color** dialog box.
6. Select or clear the **Enable assembly highlighting** check box.
7. Select or clear the **Use drill down in specific filters** check box.
8. Select a Navigation model.
9. To set alternate name parts in the structure tree:
   a. Select or clear the **Use alternate part names in the tree** check box.
   b. Click Configure. The **Configure Alternate Part Names** dialog box opens.
   c. In the first, third and fifth boxes enter text to display. In the second and the fourth boxes, select attributes from the lists.
   d. Click **OK**. The **Configure Alternate Part Names** dialog box closes.
10. Click **Apply** to apply the settings and continue setting options, or click **OK** to apply the settings and close the dialog box.

Using the Mouse in ProductView Mode

When the ProductView navigation mode is set, use the mouse in the following ways:

ProductView Navigation Mode for the Model View

Action	Result	Details
Click.	Selects a part or geometry within a part.	
Right-click and drag.	Rotates the structure.	
Press CTRL + right-click and drag.	Pans the structure.	
Press SHIFT + right-click and drag	Zooms in or out.	Drag down to zoom in, and drag up to zoom out.
Middle-click and drag.		Drag up to zoom in, and drag down to zoom out.
Spin the mouse wheel.		

ProductView Navigation Mode for the Drawing or Image View

Action	Result	Details
Right-click and drag.	Pans the view.	
Press SHIFT+middle-click and drag.		
Press SHIFT + right-click and drag.	Zooms in or out.	Drag down to zoom in, and drag up to zoom out.
Middle-click and drag.		

Using the Mouse in Creo Mode

When the Creo navigation mode is set, use the mouse in the following ways:

Creo Mode for the Model View

Action	Result	Details
Click.	Selects a part or geometry within a part.	
Right-click and drag.	Flies.	Drag down to fly out, and drag up to fly in.
Middle-click and drag.	Rotates the structure.	
Press SHIFT+middle-click and drag.	Pans.	
Press CTRL+middle-click and drag.	Zooms in or out.	Drag down to zoom out, and drag up to zoom in.

Selecting MCAD Data

About Selecting MCAD Data in the Graphics Area

You can select one or more parts or markups in the graphics area for the loaded assembly. By default, selected parts are highlighted and surrounded by a bounding box. With Creo View's smart selection function, you can refine the selection of a part to a surface or edge on that part. You can then refine the selection further:

- Point
- Endpoint
- Midpoint

Set options to enable or disable preselection highlighting for 3D parts. For other geometry, it is enabled by default. You can set a smart selection filter, or you can choose one filter to select that type of item. You can view a list of the items that are selected, and you can see a summary of the number of selected items in the status bar. You can also select or deselect all parts simultaneously.

To Set the MCAD Selection Mode

1. Click the arrow next to **Home** ▸  **Select Mode**. The selection mode menu opens.
2. On the **Home** tab, in the **Selection** group, click **Select Mode**.
3. Select one of these commands:
   -  **Select Mode**
   -  **Select All Parts**
   -  **Deselect All Parts**

About Smart Selection

You can use smart selection to select the geometry of a part. Select a part, and then place the pointer over a surface or an edge of the part to select it. When you place the pointer over a point on the selected surface or edge, you can select that point.

To Select a Point using Smart Selection

1. In the graphics area, select the part that contains the surface or edge that you want to select.
2. Move the pointer over the surface or edge. The feature is pre-highlighted.
3. Select the pre-highlighted feature.
4. To select a point on the feature, move the pointer over the point. It is pre-highlighted.
5. Select the point.

Selecting Items from the List

When you make a selection in the graphics area, all parts or annotations located under the pointer are added to a list of selectable items. These items can be accessed using these commands on the shortcut menu:

- **Select Previous**
- **Select Next**
- **Select From List**

You can also use these shortcuts to make selections:

Action	Result
Press CTRL + right-click.	Selects the next item in the list.
Press CTRL + middle-scroll.	Scrolls to another item in the list.

To Change the Selection Filter

1. On the status bar, select a filter from the list. The selection filter is changed for the active view.

To Set Selection Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **General**, select **Navigation**. The **General-Navigation** options open on the right.
3. To set the options for pre-selection highlighting of parts, surfaces, edges, and points:
   - Select or clear the **Enable pre-selection highlighting** check box.
   - To change the highlighting color, click the color box and set a new color in the **Choose Color** dialog box.
4. To set the options for highlighting of selected parts, surfaces, edges, and points:
   - Select or clear the **Select using color** check box.
   - To change the highlighting color, click the color box and set a new color in the **Choose Color** dialog box.
5. Click **Apply** to apply the settings and continue setting options, or click **OK** to apply the settings and close the dialog box.

Selecting ECAD Data

About Selecting ECAD Data

The ECAD substructure appears on the  substructure pane when Creo View ECAD is open. When you select an object from one of the partitions on the  substructure pane, the object is automatically selected in all of the partitions and in the graphics area. For most object types, you can open every partition in which the object appears. These are the exceptions:

- **Devices**
- **Part Numbers**
- **Special Symbols**

You can expand a node to select one or more objects in the node. Select an entire node in a partition to select every object associated with that node. When you select an object in a node, the related objects are also selected. These are some examples:

Example of Selected Object	Related Objects Also Selected
Net node	All associated objects
Package node	All instances of the package
Component	Device and part number
Pin	Net

You can make your selection stand out by adding a crosshair when one object is selected. When you select a component, information about it appears in the  Attributes pane of the lower data panel. You can also use cross-selection mode to select objects simultaneously in one or more of these locations:

- Multiple views
- Creo View MCAD
- Other Creo View ECAD tools

To Select Among ECAD Objects with the Same Location

1. Move the pointer over an object in the graphics area. The object is highlighted and a tooltip displays its name.
2. When multiple objects exist at a location, click and hold. A list of the objects appears.
3. In the list, move the pointer over the object name, and then release. The object is selected.

To Mark a Selected ECAD Object with a Crosshair

1. Click **Home** ▸  **Crosshair**. Crosshair selection mode is activated.
2. Select one object. The selection is indicated with a highlight and a crosshair.
3. To turn off crosshair selection mode, click **Home** ▸ **Crosshair** again.

Selectable Objects in PCB Viewables

This table shows the types of objects found in PCB viewables, and whether or not they can be selected graphically.

Object	Graphically Selectable	Comments
Board Outline	Yes	
Component	Yes	
Net	No	Only physical object associated with nets can be selected.
Via	Yes	Does not appear as a selectable object in the structure pane.
Figure	Yes	Does not appear as a selectable object in the structure pane. When selected, associated data appears in the Attributes pane.
Region	Yes	
Pad	Yes	
Padstack	No	
Pin	No	
Mechanical Hole	Yes	If the hole is a standalone hole, only the hole is selected. If the hole is part of a component definition, the component is selected.
Test Point	Yes	Does not appear as a selectable object in the structure pane.
Logical Connection (ratline/ratsnest line)	Yes	The pin-to-pin line connection is selected.
Res Des	No	Text that is part of a component is selectable as a part of the component.
Text	Yes	Text that is **not** part of an object or component.
Geometry	Yes	Geometry is part of an object, but only the geometry is highlighted.

Object	Graphically Selectable	Comments
		When selected, associated data appears in the Attributes pane.
Pin Pair	No	
Markup	Yes	

Selectable Objects in Schematic Viewables

This table shows the types of objects found in Schematic viewables, and whether or not they can be selected graphically.

Object	Graphically Selectable	Comments
Net	Yes	Selectable through a wire, junction point etc.
Symbol	Yes	Components, fractured components and special symbols are selectable. Page borders, title blocks etc. are special symbols.
Figure	Yes	Defined as special symbol.
Pin	Yes	
Junction Point	Yes	Selects the net.
Res Des	Yes	Text that is part of a component is selected as part of the component.
Text	Yes	If the text is associated with a component or a symbol, the component or the symbol is selected. Some text may not be selectable.
Geometry	No	Usually associated with a special symbol.
Pin Pair	No	Part of a net.
Markup	Yes	

Selection Mode Functionality

When you set a selection mode, subsequent selections are made according to that mode until another selection mode is selected, or until the end of the session. The default is Smart Selection.

Mode	Schematic View	PCB View	
Smart Selection	Use to select any object that does not meet more specific criteria or selection		

Mode	Schematic View	PCB View	
	options.		
Component Selection	The component symbol is selected when any of its associated object are selected in the graphics area.	The component is selected when any of its associated object are selected in the graphics area.	
Net Selection	When you select objects associated with a net, the entire net is selected and highlighted in the graphic view and the structure.		
Deselect All	Deselects all current selections		

About Filtering ECAD Selection

ECAD designs, especially PCB layouts, can have thousands of objects, often overlapping or sharing location. You can activate the selection filter in order to search for specific types of objects by selecting an option on the Status bar or by selecting an option from  **Selection Mode**.

You can use smart selection to select any object in the graphics window.

Selecting ECAD Data across Tools

About Cross-Selection Mode

In Creo View ECAD's **Cross-Selection Mode**, you can select objects in two or more open views simultaneously. Select one or more objects in the active view, and the objects are selected automatically in the receiving views. You can select one or more receiving views from among the views open and loaded in your session:

- PCB Layout
- Schematic
- MCAD model

By default, the receiving views adjust to center on the selection, though you can also choose to zoom to the selection. The active view remains unchanged. If an object cannot be found in the receiving view, a dialog box provides more information and a message appears in the lower data panel's  Messages pane.

When you activate the **External Communication** option, your receiving views can include other open sessions of Creo View, Creo View ECAD Validate, and Creo Parametric. To receive the selections in other sessions of Creo View, the **External Communication** option must also be activated there.

Creo View can also read messages from Creo View ECAD Validate and change proposal information from Creo Parametric when **External Communication** is activated.

To Cross-Select Objects

1. Click **Home** ▸ **Cross-Selection Mode**. The cross-selection menu opens.
2. Select one or more views to become receiving views.
3. To cross-select in other sessions of Creo View, other tools, or both, select **External Communication**.

   Note

   *Make sure that the **External Communication** command is also activated in the other sessions of Creo View.*
4. Select a display option for the receiving views:
   - **Center on Cross-Selection Sent**
   - **Zoom to Cross-Selection Sent**
5. Select one or more objects in the active view. The objects are selected automatically in the receiving views.
6. To toggle cross-selection mode on and off with the same settings, click **Home** ▸ **Cross-Selection Mode**.

Objects Available for Cross-Selection

From each type of viewable, you can cross-select these objects:

Viewable	Object
PCB Layout	Component Pin Via Net

Viewable	Object
	Route Segment Mechanical Hole Drill Hole Region Test Point Logical Connection Scheduled Connection Board Outline Planes Conductive Figures
Schematic	Component Pin Net Bus Hierarchical Block

Cross-Selecting Objects in Different Types of Views

When the active view type is different than the receiving view type, objects are cross-selected as follows:

Active View	Receiving View	Selection	Resulting Cross-Selection
Schematic	PCB Layout	Net	All objects associated with the net
		Pin	Pin pad
		Bus	All objects logically connected to the bus and its bits
PCB Layout	Schematic	Pin pad	Pin
		Route segment	Net
		Via pad	
		Plane and	

Active View	Receiving View	Selection	Resulting Cross-Selection
		conducting figures	
		Logical connection	
		Scheduled connection	
		Test point (via)	
		Test point (pin)	Pin

About Receiving and Displaying ECAD Images

An ECAD image is an IDX file (`*.idx`) received from Creo View ECAD Compare or Creo Parametric for the following purposes:

- Creo View ECAD Compare—Messages
- Creo Parametric—Change proposals

You can receive any number of images for each view.

The **External Communication** option in the **Cross-Selection Mode** menu must be selected to receive images. Navigation and visibility settings in Creo View ECAD Compare are also transmitted to Creo View ECAD. Control the visibility of an image using the  Color and Visibility pane of the upper data panel. The names of received images appear on this pane, under the **Images** sublayer of the **Annotations** layer. Use the  View Content pane to control image transparency or to rename or delete an image. When you apply the **Transparency** option to an image, it is displayed at 50 percent opacity. You can add markup and save the view as an annotation set.

To Receive Images

Click **Home** ▸  **Cross-Selection Mode** ▸ **External Communication**. Creo View is configured to receive images.

To Display Received Images

1. Click the  Color and Visibility pane of the upper data panel. The pane opens.
2. Expand the **Annotations** layer node, and then expand the **Images** layer node. A list of received images and grids opens.

3. Select the image's visibility check box. The image is displayed over the design.
4. To reduce the opacity of the image, perform these actions:
   a. Click the  View Content pane of the upper data panel. It opens with a list of received images.
   b. Right-click the image and choose **Transparent** from the shortcut menu. The image is adjusted.

Interpreting Images from Creo View ECAD Validate

You can receive any number of images for each view. Messages from Creo View ECAD Validate are the result of a comparison between two designs, the Current design and the Compare To design. Differences in the designs are designated in Creo View ECAD Validate as **Add**, **Modify**, or **Delete**. In Creo View, the differences do not have labels. Instead, the **Current** design becomes the view and the image displays a record of the differences:

- **Add**—Selects the added object in the design.
- **Modify**—Creates an object on the image layer and selects the original object in the design.
- **Delete**—Creates the deleted object on the image layer.

A corresponding grid sublayer is also created when an image is received from Creo View ECAD Validate. The grid identifies the regions containing differences. The grid name is the name of the image with the suffix `_grids`.

To Rename or Delete Received Images

1. Click the  View Content pane of the upper data panel. The pane opens with a list of received images.
2. To rename an image, right-click it and choose **Rename** from the shortcut menu. The **Rename** dialog box opens.
3. Type a new name and click **OK**. The image is renamed.
4. To delete an image, right-click it and choose **Delete Selected** from the shortcut menu. The image is deleted.

8

Working with Viewables

About Filtering the List of Viewables

You can filter the list of viewables for any MCAD file. Set the filter to show only viewables or parts with a certain name, or show only viewables that are associated with selected parts. You can use wildcard characters for setting the filters. You can make your settings the default.

To Filter the List of Viewables

1. In the Viewables pane of the primary panel, right-click and choose **Filter Files** from the shortcut menu. The **Filter** list dialog box opens.
2. Select one or more of the options under **Show in image list**:
   - **Names matching**
   - **Part names matching**
   - **Only files attached to selected parts**
3. To save your settings as the default, click **Make Default**.
4. Click **OK**.

MCAD Viewables

About MCAD Viewables

You can display different kinds of views of CAD objects. The **Viewables** panel contains folders for the following types of views, depending on what views are saved to the CAD object:

View	Details
 **Model**	A 3D view. You can manipulate the model and its components.
 **Image**	A 2D view
 **Drawing**	A technical plan
 **Document**	A PDF
 **Illustration**	A technical illustration. You can view animations

You can add markups to any type of viewable.

Navigating the MCAD View

Use the following commands on the **Home** tab, in the **Navigation** group, or on the **Structure** tab in the **Parts** group, to navigate:

View	Action	Result
All	Click  **Zoom All**.	Resizes the part to fit the graphics area.
Model, Image, Document, and Drawing	Click  **Zoom Window**, and then select a rectangular region of the representation.	Zooms to fit the selected region to the graphics area.
Structure, Model, Image, Document, and Drawing	Select an item in the representation, and then click  **Zoom Selected**	Zooms in to the selected item.
Model	Click  **Spin Center**, and then right-click and hold while moving the pointer.	Rotates the representation around the point you selected.
Model and Structure	Click **Orientation**, and then select an orientation from the list.	Orients the model.
Image, Drawing, and Document	Click  **Zoom In** or  **Zoom Out**.	Zooms incrementally with each click.
Image, Drawing, and Document	Click  **Fit Width** or  **Fit Height**.	Resizes the representation to the width or the height of the graphics area.
Image, Drawing, and Document	Click  **Rotate Right** or  **Rotate Left**.	Rotates the representation counterclockwise or clockwise.
Image and Drawing	Click  **Flip Horizontal** or  **Flip Vertical**.	Flips the representation.
Document	Click  **Actual Size**.	Resizes the document to its actual size.

Model View

About the Model View

You can open the Model View to see a 3D representation of a CAD design. In the Model View, you can perform the following operations:

- Set the Render Mode
- Apply Display Filters
- Change the location of components in the model
- Create a sectioned view
- Create construction geometry

The Model View is a Creo View Structure (`*.pvs`) file. When you save the file, changes to the display are also saved.

To Apply Display Filters

1. Click **Home** ▸ **Filters**. The **Display Filters** dialog box opens.
2. Select or clear the check boxes next to the items to display or hide.
3. Click **Apply** and then click **OK**.

About View States

A structure has one or more view states associated with it. A view state is a model view with one or more of the following properties set:

- Orientation
- Render mode
- Transparency
- Sectioning
- Layer state
- Load status of parts
- Markup created in Creo View
- Model Annotations
- Simplified Representations

Use a view state to display the structure with the appearance and settings of your choice. You can open one or more view states, and you can apply a new view state to the current model view. When you apply a new view state while using the Design Check tool in Creo View, your tags and tag notes are preserved.

To Apply a New View State

1. In an open view, click  . The Viewables pane of the primary panel opens.
2. To apply a view state to the current view, select it and do one of these operations:
   - Double-click.
   - Right-click and choose **Apply to Current View**.
3. To apply a view state to another view, right-click the view state and choose **Apply to View**, and then select the view.

To Change a Display with Local Display Style

1. Select a component of the representation.
2. Activate the shortcut menu and move the pointer over **Local Display Style**.
3. Select a display mode.
4. The selected component is displayed accordingly.

View Orientation

About View Orientation in Figures

A view orientation is the angle of the displayed structure in the graphics area. There are eight standard view orientations. You can add, edit, and delete view orientations in the **Custom View Orientations** dialog box. These changes are immediately implemented in the **Orientation** list. The Undo and Redo mechanism is available when updating the view orientations list. Although by default view orientations are global, you can set a view orientation relative to a coordinate system.

You can modify view orientation in Creo View, but you can only select existing view orientations in Creo View Lite or Creo View Express.

To Change the View Orientation

There are three ways to change the view orientation.

- From the Ribbon:
  1. Open a structure.
  2. On the **Home** tab in the **Navigation** group or on the **Structure** tab in the **Parts** group, click **Orientation**. The **Orientation** list opens.
  3. Click a view orientation from the list. The view orientation changes.
- From the **Custom View Orientations** dialog box:
  1. Open a structure.
  2. On the **Home** tab in the **Navigation** group or on the **Structure** tab in the **Parts** group, click **Orientation**. The **Orientation** list opens.
  3. Click **Customize**. The **Custom View Orientations** dialog box opens.
  4. Click a view orientation in the list, then either right-click and choose **Activate Orientation** from the shortcut menu, or click  on the **Custom View Orientations** dialog box. The view orientation changes, but the **Custom View Orientations** dialog box remains open, so you can repeat this step to compare view orientations.
- From the keyboard shortcuts in some of the standard view orientations: Click in the graphics area, and then press the keyboard shortcut letter that appears on the **Orientation** list.

To Add a New View Orientation

1. Open the **Custom View Orientations** dialog box.
2. To open the **Add New Orientation** dialog box do one of the following actions:
   - Click **New**.
   - Right-click in the **Custom View Orientations** dialog box, and then choose **New Orientation**.

   The coordinate values of the open structure appear in the boxes.
3. Type a name for the new view orientation.
4. Modify the values in the boxes.
5. Click **Activate** to orientate the display in the graphics area to the new values.

   Note

   *You can click **Reset Values** to return the rotation values to the initial state.*
6. Click **OK**. The new view orientation is added to the list for immediate use.
7. Click **Close**.

To Change the Default View Orientation

1. Open the **Custom View Orientations** dialog box.
2. Right-click a view from the list and choose **Set as Default**. The selected view is now boldfaced, indicating that it is default.
3. Click **Close**.

To Change the Order of the View Orientation List

1. Open the **Custom View Orientations** dialog box.
2. To change the row position in the view orientation list do one of the following actions:
   - Select a view orientation from the list and click the **Move Up** or **Move Down** arrow.
   - Select a view orientation from the list, right-click and choose **Move Up** or **Move Down** arrow.
   - Select a view orientation from the list, and drag it to the required position.
3. To set a view orientation at the beginning or end of the view orientation list, right-click a view orientation and choose **Move To Top** or **Move to Bottom**.
4. Click **Close**.

To Delete View Orientation

1. Open the **Custom View Orientations** dialog box.
2. Delete a view orientation in one of the following ways:
   - Select a view orientation from the list and click **Delete**.
   - Select a view orientation from the list and press DELETE.
   - Right-click a view orientation and choose **Delete**.
3. Click **Close**.

To Edit View Orientation

1. Open the **Custom View Orientations** dialog box.
2. Right-click a view in the **Custom View Orientations** dialog box and then choose **Edit**, or select a view from the list and click **Edit**. **Edit View Orientations** dialog box opens.
3. Modify the name or the values in the boxes, and then click **Activate**. The display in the graphics area orientates to the new values.

Note

*You can click **Reset Values** to return the rotation values to the initial state.*

4. Click **OK** and **Close**.

Opening the Custom View Orientations Dialog Box

Open the **Custom View Orientations** dialog box in one of the following ways:

- Click in the graphics area, and then press U.
- From the Ribbon:
  1. On the **Home** tab in the **Navigation** group or on the **Structure** tab in the **Parts** group, click **Orientation**.
  2. Click **Customize**. The **Custom View Orientations** dialog box opens.

To Set a View Orientation Relative to a Coordinate System

1. Open a structure with a coordinate system.
2. Select a coordinate system.
3. Right-click and select **Relative View Orientation**.
4. Select a view orientation from the list. The view orientation is displayed relative to the selected coordinate system.

To Restore Original View Orientations

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, click **Global**.
3. On the left, click **Model**. The **Model-Main** options open on the right.
4. Click **Restore to Original**. A message opens, warning you that you will delete all customized orientations and restore the original ones.
5. Click **Yes** and **OK** to continue.

Drawing View

About the Drawing View

You can view drawings of various file types, and you can annotate a drawing and save it. Configure Creo View and Creo View Lite in the following ways to read drawing files:

- HPGL—Define pen maps for viewing and printing
- DXF and DWG—Select and manage fonts
- AutoCAD—Control the visibility of layers and blocks

If you install Creo View from the Windchill Software Downloads page, you must select the AutoCAD Viewers option to open DXF and DWG files. You can compare two drawings. Use the option settings to set the tolerance for arcs and to modify the drawing viewer display.

To View Drawings with Multiple Sheets

1. Make sure that a drawing with two or more sheets is open.
2. To scroll through the sheets, do one of the following operations:
   - Click one of these buttons:
     - **Home** ▸  **First**
     - **Home** ▸  **Previous**
     - **Home** ▸  **Next**
     - **Home** ▸  **Last**
   - In the graphics area, right-click and choose **Next Sheet** or **Previous Sheet** from the shortcut menu.
3. To jump directly to a sheet, do one of the following operations:
   - On the **Home** tab, in the **Sheets** group, type the page number in the box and press ENTER.
   - In the graphics area, right-click and choose **Go To Sheet** from the shortcut menu. In the **Go To Sheet** dialog box, type the sheet number in the **Sheet** box and click **OK**.

About Comparing Drawings

In Creo View, you can compare 2D drawings to determine areas that overlap as well as differences. You can specify how to align the drawings for comparison, and there are many options for specifying comparison parameters. You can use this feature, for example, to compare two versions of the same drawing to identify the differences. Once you've selected the two drawings, aligned them properly, and enabled comparison, the areas which are common will display in black, with the areas specific to drawing A in red and areas specific to drawing B in green.

To Compare Two Drawings

1. In the **Viewables** tab, do one of the following operations:
   - Right-click a drawing in the list, choose **Compare With**, and select the second drawing for the comparison.
   - Select two drawings from the list, right-click, and choose **Compare Selected**.
2. Set one or more of the following commands on the **Compare** tab:
   - **Align by Center** aligns the two drawings by lining up the center point of each drawing.
   - **Align by References** lets you select two reference locations (one on the first drawing and one on the second drawing) by which to align the drawings. This option displays the **Select Alignment References** window, where you can either click in the drawing view to populate the **Reference A** and **Reference B** fields to specify alignment anchors, or click **CSYS** to specify a coordinate system for each field.
   - **Show Both** displays drawing A in red, drawing B in green, and their intersection in black.
   - **Show A** displays only drawing A in red.
   - **Show B** displays only drawing B in green.
   - **Show A-B** displays drawing A in red, minus the portion of that drawing that intersects with drawing B, which appears in black.
   - **Show B-A** displays drawing B in green, minus the portion of that drawing that intersects with drawing A, which appears in black.
   - **Show Overlap** displays only the area that overlap between drawing A and drawing B.
   - The **Drawing** menu lets you select which drawing the sheet control will apply to.
   - **Previous**, **Next**, **First**, and **Last** sheets help you move through multi-sheet drawings. You can also specify a sheet number, such as **Sheet** 2 of 5, to display.

To Set General Drawing Viewer Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, select **Drawing Viewer**. The **Drawing Viewer-Main** options open on the right.
3. Under **Load Settings**, set the following options:

- **Approximate arcs from polylines**—Polylines are analyzed to determine whether they are arcs, and can then be used in radius measurements.
- **Tolerance**—Type a number from `0.1` to `0.75`.

4. To set options for the active view, in the **Showing** box, select **Active View**. The **Active View** options open.
5. On the left, select **Drawing Viewer**. The **Drawing Viewer-Main** options open on the right.
6. To configure display colors, select or clear the **Monochrome display** check box.
7. Under **Color**, select or clear the **Use background color from drawing** check box.
8. To set a new background color, follow these steps:
   a. Click the color button. The **Choose Color** dialog box opens.
   b. Under **Basic colors**, select a color, or under **Settings**, select values for red, green and blue.
   c. Click **OK**. The **Choose Color** dialog box closes.
9. In the **Creo View Options** dialog box, click **OK** to apply the changes and close the dialog box, or click **Apply** to apply the changes and keep setting options.

Setting the Tolerance for Arcs in Drawings

The tolerance setting is used for fitting arcs to polylines. If the difference in radius between the two arcs falls within the tolerance you set, the arcs are merged. The tolerance is a percentage: when you set a value of `0.15`, the tolerance is set at 15 percent.

Modifying the Display in Drawing View

Perform these actions to modify the display of a drawing:

Action	Result
Click **Home** ▸  **Filters** ▸ **Markups**.	Markups are displayed or hidden.
Click **Markup**, and then type a number in the **Drawing Scale** box.	Adjusts the scale.

About Pen Maps

In Creo View and Creo View Lite, a pen map is a collection of drawing viewer option settings selected to display an HPGL drawing correctly in the graphics area and when printing. In the Global options, you can configure one or more pen maps and make your settings the default. In the Active View options, you can select a pen map to apply to a selected view.

To Configure the HPGL Pen Map

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **Drawing Viewer**, click **HPGL**. The **Drawing Viewer-HPGL** options open on the right.
3. Under **Pen Mapping**, select the **Use pen map** check box.
4. To add a pen map, click **Add**. The **HPGL Pen Map** dialog box opens.
5. In the **Name** box, type a name for the pen map.
6. Click **Add**. The **Choose Color** dialog box opens.
7. Select a color and click **OK**. The pen map is added to the list.
8. To change the line width, double-click the default value in the **Line Width** column of the list. The **Line Width** dialog box opens.
9. Type a new value and click **OK**.
10. Repeat steps 7–10 for each pen map that you want to create.
11. Click **OK**. The set of pen maps is added to the list.
12. Click **OK** to apply the changes and close the **Creo View Options** dialog box, or click **Apply** to save the changes and continue setting options.

About Drawing Layers and Blocks

Drawing files from AutoCAD can include layers and blocks. Show or hide layers and blocks to change the drawing display. You can save your visibility settings as an annotation set.

To Show or Hide Drawing Layers

1. In an open drawing, in the upper data panel, click  . The Layers pane opens.
2. Under **Layers**, select the check box next to one or more layers to show, and clear the check box next to one or more layers to hide. The layer visibility is updated automatically.
3. To revert to the original visibility settings, select a layer that is modified, right-click, and choose a command from the shortcut menu:
   - **Reset to Original**—Resets the selected layer.
   - **Reset All to Original**—Resets all modified layers.
4. To save the settings, follow these steps:
   a. Click **File** ▸ **Save Annotation Set As**. The **New Annotation Set** dialog box opens.

b. Type the annotation set name details in the boxes and click **OK**. The layer settings are saved.

To Manage Fonts for DXF or DWG Files

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **Drawing Viewer**, click **DXF - DWG**. The **Drawing Viewer-DXF - DWG** options open on the right.
3. To add a new font file path, follow these steps:
   a. Click **Add**. The **Add Font Search Path** dialog box opens.
   b. Click .... The **Select DXF Font Directory** dialog box opens.
4. To edit a file path, follow these steps:
   a. Under **Font**, select the file path.
   b. Click **Edit**. The **Edit Font Search Path** dialog box opens.
5. Click .... The **Select DXF Font Directory** dialog box opens.
6. Browse to and select the file path of the font, and then click **Select DXF Font Directory**.
7. In the dialog box, click **OK**. The path appears in the **Font** box.
8. To delete a file path, under **Font**, select it and click **Delete**.
9. Select or clear the **Use big font** check box.
10. Click **OK** to apply the changes and close the **Creo View Options** dialog box, or click **Apply** to save the changes and continue setting options.

To Load the Frozen Layers of Drawing File

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **Drawing Viewer**, click **DXF - DWG**. The **Drawing Viewer- DXF - DWG** options open on the right.
3. Under **Layers**, select the **Load Frozen Layers** check box.
4. Click **OK** to apply the changes and close the **Creo View Options** dialog box, or click **Apply** to save the changes and continue setting options.

To Show or Hide Drawing Blocks

1. In an open drawing, in the upper data panel, click  . The Blocks pane opens.
2. Under **Blocks**, select the check box next to one or more blocks to show, and clear the check box next to one or more blocks to hide. The block visibility is updated automatically.

3. To revert to the original visibility settings, select a block that is modified, right-click, and choose a command from the shortcut menu:
   - **Reset to Original**—Resets the selected block.
   - **Reset All to Original**—Resets all modified blocks.
4. To save the settings, follow these steps:
   a. Click **File** ▸ **Save Annotation Set As**. The **New Annotation Set** dialog box opens.
   b. Type the annotation set name details in the boxes and click **OK**. The block settings are saved.

Image View

Image Pages Commands

On the **Home** tab in the **Sheets** group, use **Previous**, **First**, **Next**, and **Last**, as well as the current sheet number, to navigate multi-page images.

Modifying the Display in Image View

On the **Home** tab, in the **Display** group, use the following commands to modify the display in an image view:

Action	Result
Click **Filters** ▸ **Markups**.	Markups are displayed or hidden.
Click **Invert Colors**.	The image colors are inverted.

Note

*You can also right-click the graphics area and choose **Invert Colors** from the shortcut menu.*

Rotating and Flipping an Image

Use the following commands to rotate an image:

- On the **Home** tab, in the **Navigation** group, click **Rotate Left** or **Rotate Right**.
- In the graphics area, right-click and choose **Rotate or Flip** ▸ **Rotate Clockwise** or **Rotate or Flip** ▸ **Rotate Counterclockwise**.

Use the following commands to flip an image:

- On the **Home** tab, in the **Navigation** group, click **Flip Horizontal** or **Flip Vertical**.
- In the graphics area, right-click and choose **Rotate or Flip** ▸ **Flip Vertical** or **Rotate or Flip** ▸ **Flip Horizontal**.

Document View

About Viewing Documents

You can open documents from the  Document partition of the  Viewables pane in Creo View Lite and Creo View. While the PDF viewer is embedded in Creo View, other document formats open in their appropriate programs.

Use the  **Find** tool to search for text in PDFs. You can navigate, view markups, and print PDF files. You can add markups to any PDF as long as either Adobe Standard or Adobe Professional is installed. If you have only Adobe Reader then you can only make markups when viewing a PDF with markup rights enabled.

You can view PDF drawings in Creo View, but you cannot create measurements or construction geometry on them, or convert them to 2D vector files or illustration files.

To View Documents with Multiple Sheets

1. To scroll through the sheets, do one of the following operations:
   - Click one of these buttons:
     - **Home** ▸  **First**
     - **Home** ▸  **Previous**
     - **Home** ▸  **Next**
     - **Home** ▸  **Last**
   - In the graphics area, right-click and choose **Next Page** or **Previous Page** from the shortcut menu.
2. To jump directly to a sheet, do one of the following operations:
   - On the **Home** tab, in the **Pages** group, type the page number in the box and press ENTER.
   - Click a page number in the  **Pages** pane on the  **View** pane on the primary panel.
   - Click an item in the  **Bookmarks** pane on the  **View** pane on the primary panel.

To Toggle the Display of a Document's Markups

Click **Home** ▸  **Filters** ▸ **Markups**.

To Search PDFs Using the Find Tool

1. Click **Home** ▸ **Find**. The **Find** dialog box opens.
2. In the **Find** box, type the text to search for.
3. To return only exact matches, select the **Match whole word only** check box.
4. To make the find case-sensitive, select the **Match case** check box.
5. Do one of the following:
   - To find each instance of a name, click **Find Next** or **Find Previous**.
   - To find all instances of a specific name at one time, click **Find All**.

   The matching text is highlighted in the document.

Illustration View

About Illustration Views

You can use Creo View to view and mark up technical illustrations created with Arbortext IsoView. You can perform the following operations that are specific to illustration views:

- Navigate the illustration
- Play an animation
- Create or delete annotations
- Show or hide Hotspots
- Magnify an area of the illustration
- Use a **Global View** window for navigation
- Save the view as an annotation set

Illustration views are located in the **Illustration** folder in the **Viewables** pane. Annotation sets for an Illustration view are located in the **Illustration** folder in the **Annotations** pane.

To View an Illustration

1. Open a Windchill representation that contains `*.iso` illustrations.
2. In the primary panel, on the Viewables pane, click **Illustration**. The illustrations associated with the representation are listed.
3. Select an illustration. IsoView opens, and the illustration opens in the graphics area.

To View an Animated Illustration

1. Open a representation that contains `*.iso` animated illustrations.
2. In the primary panel, on the Viewables pane, click **Illustration**. The illustrations associated with the representation are listed.
3. Select an illustration. IsoView opens, the illustration opens in the viewing area, and the animation begins.
4. To control the animation playback, on the ribbon, click **Animation**. The **Animation** tab opens.
5. In the **Animation** group, select one of the following commands:
   - **Play**
   - **Stop**
   - **Rewind**

To View an Illustration with the Magnifier Tool

1. Open an illustration view.
2. On the **Home** tab, in the **Display** group, click **Magnifier**. The **Magnifier** window opens.
3. In the **Magnifier** window, move the slider up to increase the zoom, or move the slider down to decrease the zoom.
4. In the graphics area, move the pointer over the illustration. The magnified view appears simultaneously in the **Magnifier** window.

To Navigate an Illustration using the Global View Window

1. Open an illustration view.
2. On the **Home** tab, in the **Display** group, click **Global View**. The **Global View** window opens.
3. In the graphics area, pan or zoom around the illustration. In the **Global View** window, a red frame indicates the area of the model shown in the graphics area.

Using the Magnifier and the Global View

You can see a close-up of an area of the model using either of the following tools:

- **Magnifier**—You can set the level of zoom in the **Magnifier** window, and then move the mouse around the graphics area. The magnified area appears in the **Magnifier** window.
- **Global View**—You can zoom in on an area of the representation in the graphics area, and then use the display in the **Global View** window to indicate the area of

the illustration that you are viewing. The frame of the graphics area is indicated in the **Global View** window by a red frame.

To Search an Illustration

1. Make sure an illustration file is open.
2. On the **Home** tab, in the **Selection** group, click **Search** ▸ **IsoView Search**. The **Search** dialog box opens.
3. Select one of the following operations:
   - **Search for**—Finds one of the following elements:
     - **Object info**
     - **Object ID**
     - **Object name**
     - **Object tip**
     - **Object attribute**
     - **Text**
   - **Remove Highlight**—Removes the search result highlighting.
4. Select a value to find. If you select **is equal to** or **contains**, type a value in the box.
5. Click **OK**. The results are highlighted in the viewing area.

To Show or Hide Markups for Illustrations

1. Make sure an illustration file is open.
2. On the **Home** tab, in the **Display** group, click **Filters**.
3. To display markups, select **Markups**, or to hide them, clear the **Markups** option.

To Show or Hide Illustration Hotspots

1. Make sure an illustration file is open.
2. To show Hotspots, do one of the following operations:
   - On the **Home** tab, in the **Display** group, click **Hotspots**.
   - In the graphics area, right-click and choose **Show Hotspots**.
3. To hide Hotspots, do one of the following operations:
   - On the **Home** tab, in the **Display** group, click **Hotspots**.
   - In the graphics area, right-click and deselect **Show Hotspots**.

To Set Illustration View Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**.
3. In the left pane of the dialog box, select **Illustration Viewer**. The **Illustration Viewer-Main** options open in the right pane of the dialog box.
4. Under **Annotations**, set any of the following annotation options:
   - **Line Width**
   - **Color**
   - **Font Size**
5. Under **General Display**, select the **Use Anti - Aliasing** option to minimize line distortion.
6. Under **Hotspots**, select or clear the **Show alert message when Hotspot link is not available in Creo View** check box.
7. Click **Apply** to apply the changes and continue setting options, or click **OK** to apply the changes and close the dialog box.

   Note

   When you change illustration options, the changes to the appearance are applied to past illustration files as well.

Requirements for Viewing Illustrations

To view an `*.iso` illustration file, you must meet the following system requirements:

- You must use a Windows 32–bit machine.
- IsoView 7.1 must be installed and licensed.
- Illustration files must be opened from Windchill. You can only view illustrations from Windchill using Creo View.

ECAD Viewables

About ECAD Viewables

In Creo View and Creo View Lite you can display schematic and PCB layout views. The Creo View ECAD file format for these files is EDA (`*.eda`). An ECAD viewable may be a separate file, or one or more viewables may be part of a design package that also includes MCAD viewables. When you open an ECAD viewable in Creo View, a  substructure pane opens in the primary panel. It

contains all the objects in the file, organized into partitions by type. You can expand and collapse the partition labels. Objects can appear in more than one partition on the substructure tab. For instance, a pin may be included in the Components partition and under its net in the Nets partition. Next to each object the following information appears in brackets, depending on the view type:

- PCB Layout—Location on the circuit board
- Schematic—Sheet where the object appears

The  Color and Visibility pane opens in the upper data panel when you open an ECAD viewable. For each viewable, a layer is added for object types that are applicable to the whole design or that are located on more than one layer. In PCB layout views, the **Global** layer lists these objects, and in schematic views, they are on the **Schematic Objects** layer. You can use the color and visibility commands on these layers to assign a common display status to all objects of an object type. The **Annotations** layer contains the overlays or images associated with the viewable.

In the lower data panel, you can use the  Attributes pane to show some or all of the attributes of the design.

You can add markups to a schematic or PCB layout view. You can save the view in these file formats:

- Structure (PVS or PVZ)
- ECAD viewable (EDA)
- Annotation set (AST) (Creo View ECAD only)

To Show an Object in All Partitions of the Substructure Pane

In the  substructure pane, right-click one or more objects, and choose **Show In Other Partitions** from the shortcut menu.

To Expand or Collapse All Nodes of a Partition

1. Select any node in the partition. The node expands.
2. Right-click and choose **Expand All** or **Collapse All** from the shortcut menu.

Zoom for ECAD Viewables

Perform these actions to modify the zoom of the ECAD view:

Action	Result
Click **Home** ▸  **Zoom All**.	Zooms out to the maximum extents of the viewable for the sheet. ***Note*** *The maximum extents may be far beyond the outlines of the sheet.*
Click **Home** ▸  **Zoom Window**.	Zooms in to the window you define
On the  substructure pane, double-click one or more objects.	Zooms in or out to include the selected objects. ***Note*** *For items on multiple sheets, the zoom is performed on the first sheet with a selection.*
Click **Home** ▸  **Zoom Selected**.	
Select one or more objects, and then right-click and choose **Zoom Selected** from the shortcut menu.	
Select one or more object, and then right-click and choose **Center on Pick** from the shortcut menu.	Centers the view around the selected objects.
In an ECAD schematic view, click **Home** ▸ **Center on Selected**. In a PCB layout view, click **Home** ▸ **Center**.	
Click **Home** ▸  **Zoom Outline**.	Zooms in or out to the outlines of the sheet or PCB outline.

About PCB Layout Views

You can display PCB layout views in Creo View and Creo View Lite. This view shows the placement of objects in an ECAD design. The designs are made up of layers. Some of the layers, such as the **Global** layer, are generated by Creo View, while others are imported from the ECAD tool. The following layers may appear in Creo View, depending on your ECAD data:

- **Miscellaneous**
- **Annotations**

Within each layer are one or more sublayers. The sublayers contain information specific to the design, such as pads, vias, routes, and other geometries associated with the conductive pattern. Use these operations to modify the view:

- Zoom
- Change **Orientation**

- Rotate
- **Flip Sides**

To Rotate, Flip, or Orient a PCB Layout View

1. To rotate the view click **Home** ▸  **Rotate Left** or **Home** ▸  **Rotate Right**. The view is rotated by a 90–degree increment.
2. To flip the view, click **Home** ▸  **Flip Sides**. The view is flipped along a vertical axis.
3. To change the view's orientation, click **Home** ▸ **Orientation** ▸ **Top** or **Home** ▸ **Orientation** ▸ **Bottom**.

About Schematic Views

A schematic view shows the elements of the ECAD design using symbols to represent the components and their circuits. You can display schematic views in Creo View and Creo View Lite. The view contains standard sublayers on the  Color and Visibility pane. Schematic files can include several sheets. You can scroll through the sheets or select a sheet name or number to view it. When a schematic contains connectors to objects on other pages, you can jump to those pages using the shortcut menu.

In addition to the partitions for standard symbols, the  substructure pane for a schematic view contains a partition for special symbols. Special symbols represent logic and connectivity on the schematic but do not represent physical components or conductors. For example, a special symbol could be voltage supply lines connected to component symbols or pins or terminations of wires to or from another sheet of the schematic.

You can open the hierarchical blocks of a schematic in the current view or a new view using the shortcut menu.

To View a Schematic File with Multiple Sheets

1. To scroll through the sheets, click any of these buttons:
   - **Home** ▸  **First**
   - **Home** ▸  **Previous**

- **Home** ▸  **Next**
- **Home** ▸  **Last**

2. To jump directly to a sheet, do one of these operations:
   - Click **Home** ▸  **Sheet Name**, and then select a sheet from the list.
   - On the **Home** tab, type the page number in the box and press ENTER.
3. To view the other side of an off-page connector, right-click it and choose the sheet name or number from the shortcut menu.

To Open a Hierarchical Block

Right-click the block and choose one of these commands from the shortcut menu:

- **Open Block**
- **Open Block in New View**

The block is displayed.

Displaying ECAD Designs

Color and Visibility

About Design Layer Color and Visibility

In an ECAD design, a layer's appearance consists of these factors:

- Color
- Visibility status
- Position in the display order

You can control the appearance of design layers using the  Color and Visibility pane of the upper data panel. This pane appears when you open a PCB layout or schematic view and shows the layers and sublayers of a design. Use the appearance commands in these ways:

- Assign colors to layers to make them quickly identifiable.
- Show or hide layers to simplify the view.
- Set default display options in the **Creo View Options** dialog box, or make your session settings the default.

A layer can have sublayers of different colors, visibility statuses, or both. You can reuse a color any number of times for multiple layers. You can also assign one color to a group of like sublayers and select colors for groups.

Layers are displayed one on top of the other. Use the **Bring to Top** command to change the order of the display. To preview a change in layers, place the pointer over a layer to show it on top. The previous top layer becomes the second layer, and the remaining layers move downward. You can undo your changes and revert to the default order. In PCB layout views, perform these additional operations to control the appearance of the design:

- Invert the layers
- Fill visible, solid shapes

To Change the Color and Visibility of a Layer

1. On the  Color and Visibility pane of the upper data panel, select the check box for the layers that you want to display and clear the check boxes for the layers to hide.
2. To change the color of a layer or sublayer, double-click the category name and select a new color in the **Choose Color** dialog box.

   Note

   - *You can also right-click the layer and choose **Assign Color** from the shortcut menu to open the **Choose Color** dialog box.*
   - *A  mixed state indicator for a layer's color or visibility means that the sublayers have different statuses.*
3. Right-click the  Color and Visibility pane and choose an action from the shortcut menu:
   - **Apply to all**—Applies the visibility setting for a sublayer to all sublayers with the same name
   - **Save as defaults**—Makes your display settings the default
   - **Reset to defaults**—Reverts to the default display settings
4. To define the appearance of overlapping layers, click **Home** ▸ **Overlap Color**.

Example: Applying Color and Visibility to Like Sublayers

Many ECAD designs have sublayers with the same name on multiple layers. For example, more than one layer can contain a sublayer for via pads. When you right-click one of the via pads sublayers and choose **Apply to all Via pads**, the visibility settings for the sublayer are applied to every sublayer in the design named `Via pads`.

Displaying Holes and Groups

By selecting or clearing the visibility check box and setting a color on the **Global** layer, you control the display of these types of holes:

- Mechanical holes
- Holes that penetrate all layers of the circuit board

The display of non-through-hole vias depends on the display status of the layer on which they reside. You can control the display of holes for vias and pins separately. The display of groups may also depend on the status of the layers. When a group is marked visible, all of its members are visible. When a group is marked hidden, the display of its members is controlled by the visibility of the layers where the members reside.

To Display ECAD Regions, Areas, and Lightweight Holes in the Model View

1. In an open model view, click **Home** ▸  **Filters**. The **Display Filters** dialog box opens.
2. Under **Model Construction Geometry**, select the **Datum Curves** check box.
3. Click **OK**. Regions and lightweight holes are visible.

To Change the Background Color or the Group Overlap Color

1. In an ECAD Schematic or PCB Layout view, click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Active View**.
3. In the left pane of the dialog box, under **PCB Layout Viewer** or **Schematic Viewer**, select **Color and Visibility**. The **PCB Layout Viewer- Color and Visibility** or the **Schematic Viewer-Color and Visibility** options open in the right pane of the dialog box.
4. Click the button next to **Background Color** or **Group Overlap Color**. The **Choose Color** dialog box opens.
5. To change the color, do one of these operations:
   - Select a new color
   - Drag the sliders next to **Red**, **Green**, and **Blue**.
   - Type values in the boxes next to **Red**, **Green**, and **Blue**.
6. For PCB Layout you can set the **Dimming Level** by moving the slider or by adjusting the value in the box.
7. Click **Apply** to apply the preferences and keep the **Creo View Options** dialog box open, or click **OK** to apply the settings and close the dialog box.

Displaying Overlapping ECAD Layers

The status of the  **Overlap Color** button determines the display of two or more layers at their points of overlap:

- On—Blends the colors of the layers
- Off—Covers the bottom layers with the top layer

By default, the button is off.

To Set Color and Visibility Defaults

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Defaults**.
3. On the left, under **PCB Layout Viewer** or **Schematic Viewer**, select **Color and Visibility**. The **Color and Visibility** page opens.
4. Select the objects in the list to show by default. Clear the check boxes for the objects to hide.
5. To set a default color for an object type, double-click it, and then select a color in the **Choose Color** dialog box.
6. Click the color box to change the default color for these elements of the design:
   - **Background Color**
   - **Group Overlap Color**
7. To set the brightness of colors in a PCB Layout view, move the **Dimming Level** slider or adjust the brightness percentage in the box.
8. Click **OK** .

To Change the Display Order of Layers

1. Click **Home** ▸ **Bring to Top**. A list of the layers appears.
2. Select a layer.

   Note

   - *You can only bring a design layer to the top.*
   - *In schematic views,* ***Pins*** *is the default top layer.*
3. To undo your changes, click **Home** ▸ **Bring to Top** ▸ **Reset to Default**. The original order is restored.

To Invert the Display of Layers

In a PCB Layout view, click **Home** ▸ **Invert Layers**. The display of the layers is inverted.

To Toggle the Fill of Shapes in PCB Layout Views

Click **Home** ▸  **Fill Shapes**. All visible, solid shapes are filled or only the outlines of shapes are shown.

Highlighting and Isolating Objects

About Highlighting and Isolating Objects

In Creo View ECAD you can use the following tools to emphasize selected objects in a PCB Layout view:

-  **Isolate**—Dims the surrounding objects and displays the selected objects at full intensity
-  **Highlight**—Applies a different color on top of the selected objects. It does not change the underlying color and visibility settings for the object type

You can highlight these objects:

- Components
- Nets
- Pins
- Test points
- Regions
- Mechanical holes

Apply a solid highlight to cover the objects' color, or select a dashed highlight to keep the color visible underneath. When one or more highlighted objects overlap, their display is controlled by the  **Overlap Color** setting. You can refine selections to highlight or isolate by performing these actions:

- Limit your selection to visible sublayers or allow the selection of related geometries on other sublayers.
- View and edit a list of the highlighted or isolated objects to modify the display.

In schematic views, only the  **Highlight** tool is available. When you highlight an object in a schematic view, all of the associated objects, such as lines, symbols, and text, are also highlighted. When you highlight a fractured component or a net that spans multiple sheets, the first instance is shown. When you highlight objects on an undisplayed sheet, you must navigate to the sheet to view the highlighting.

In Creo View Lite these tools are read-only.

To Highlight Objects

1. Click **Home**, and then click the arrow next to **Highlight**. The **Highlight** menu opens.
2. To define the objects available for highlighting in PCB Layout views, select one of these options:
   - **Only Visible Sublayers**
   - **All Sublayers**
3. To set the highlight color, perform these actions:
   a. Click **Home** ▸ **Highlight** ▸ **Define Highlight Color**. The **Choose Color** dialog box opens.
   b. Select a color and click **OK**.
4. To set the highlight style, click **Home** ▸ **Highlight**, and then select or deselect the **Dashed Highlight** option.
5. In the graphics area or on the substructure tab, select one or more objects to highlight.
6. Click **Home** ▸ **Highlight** ▸ **Highlight**. The selected objects are highlighted.

   Note

   *You can also right-click the graphics area and choose **Highlight** ▸ **Highlight** from the shortcut menu.*

To Add Highlights to Additional Objects

Select the objects to highlight, and then click **Home** ▸ **Highlight** ▸ **Add Highlight**. The additional objects are highlighted.

Note

- *Clicking **Add Highlight** adds to the existing highlights, while **Highlight** removes existing highlight and highlights the selected objects.*
- *You can also right-click the graphics area and choose **Highlight** ▸ **Add to Highlight** from the shortcut menu.*

Highlighting Buses

When you select a bus trunk to highlight, the entire bus and all its associated nets are highlighted. When you highlight a net associated with a bus, the net's full path and the bus trunk are highlighted.

To Clear Highlighting

You can clear highlighting in two ways:

- To remove all highlighting, click **Home** ▸  **Highlight** ▸ **Clear All Highlights**.
- To clear the highlighting for one or more objects, select them and click **Home** ▸  **Highlight** ▸ **Remove Highlight**. The highlighting is removed.

  Note

  You can also perform these actions:

  1. *Click **Home** ▸  **Highlight** ▸ **Show Highlight List**. The **Highlighted Objects** dialog box opens.*
  2. *Select one or more objects and click **Remove**.*

To Isolate Objects

1. Click **Home**, and then click the arrow next to  **Isolate**. The **Isolate** menu opens.
2. To define the objects available for isolation, select one of these options:
   - **Only Visible Sublayers**
   - **All Sublayers**
3. To set the dimming level for the rest of the objects in the design, perform these actions:
   a. Click **Home** ▸  **Isolate** ▸ **Dimming**. The **Creo View Options** dialog box opens.
   b. Next to **Dimming Level**, adjust the slider between **Dim** and **Bright**, or select a brightness level from the box.
   c. To apply your setting to other views or to the default setting, select one or both of these options:
      - **Update Defaults**
      - **Update All Views**
   d. Click **OK**. The dimming level is set.
4. To toggle the display of logical connections for isolated routes, click **Home** ▸  **Isolate**, and then select or clear the **Show Logical Connections** option.
5. Select one or more objects to isolate, and then click **Home** ▸  **Isolate** ▸ **Isolate**. The objects are isolated.

Note

The  ***Isolate*** *button removes any previous highlighting on the selected objects.*

To Clear Isolation for One or More Objects

1. To remove all isolation, click **Home** ▸  **Isolate** ▸ **Clear All Isolated Objects**.
2. To clear the isolation for one or more objects, select them and click **Home** ▸  **Isolate** ▸ **Remove Isolated Object**. The isolation is removed.

   Note

   You can also perform these actions:

   a. *Click* ***Home*** ▸  ***Isolate*** ▸ ***Show Isolated Object List****. The* ***Isolated Objects*** *dialog box opens.*

   b. *Select one or more objects and click* ***Remove****.*

Displaying Text

About Setting Text Data Options for PCB Layout Views

You can set the font style and the font size in PCB Layouts to prevent the text from being illegibly small or disproportionately large.

To Set the Option for PCB Layout Text Data

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Defaults** or **Active View**.
3. On the left, under **PCB Layout Viewer**, select **Text Data**. The **Text Data** options open.
4. In the **Default Font** box scroll to the required font.
5. In the **Font Size** box scroll to the required value.
6. Click **Apply** to apply the settings and keep the **Creo View Options** dialog box open, or click **OK** to apply the settings and close the dialog box.

Adding Overlays

About Overlays

An overlay is an image that represents the following information, depending on the file format:

- Gerber—Manufacturing artwork. A Gerber RS-274X file contains settings for placement, format, scale, and other parameters, while a Gerber RS-274D file does not.
- DXF—Mechanical or fabrication drawing

Use an overlay to compare manufacturing output data by placing the overlay over the corresponding PCB layout design. You can set the format, unit, polarity, placement, and scale for files without embedded settings. Then, load an overlay to a design in either of these ways:

- Create an overlay from an external file.
- Open an overlay viewable contained in the PVS or PVZ file.

Creo View ECAD contains commands for aligning and rotating the overlay to the design.

Note

Rotating an overlay before performing an alignment operation produces the best result.

After you have placed an overlay on a design, you can add markup. Save the overlay as an annotation set. You can also open a Gerber or DXF file in a new view. You can load more than one overlay to a design, but you can only turn on the visibility of one overlay at a time. Loaded overlays appear under **Annotations** on the  Color and Visibility pane of the upper data panel. Select any of the following types of display for an overlay:

- Opaque
- 50 percent transparent
- Opaque with an  **Overlap Color** where the geometries of the overlay and the design coincide.

You can view the tooltip of an object that is below the visible overlay if the **Exclude Overlays** option in **PCB Layout Viewer-Tooltips** is selected. If the option is not selected, you can view the tooltip of the overlay.

To Load an Overlay from the Structure

1. In an open design, click **Overlays**, and then click the arrow below **New Overlay**. The **New Overlay** list opens.
2. Select an overlay from the list. It opens in the graphics area.

To Load an Overlay from an External File

1. In an open design, click **Overlays** ▸ **New Overlay** ▸ **Load from file**. The **Open File** dialog box opens.
2. Select a file and click **Open**. The overlay is loaded.
3. To save the overlay to the structure, save it as an annotation set.

To Align an Overlay by Aligning Vertices

1. Click **Overlays** ▸  **Align**. The **Select Alignment Vertices** dialog box opens.
2. Click the **Overlay Vertex** reference collector to activate it.
3. In the graphics area, select a vertex in the overlay. It appears in the reference collector.
4. Select the corresponding vertex in the design. It appears in the **Design Data Vertex** reference collector.
5. Click **OK**. The overlay is aligned to the design.

To Align an Overlay by Rotating or Flipping

Click **Overlays**, and then select one or more commands:

-  **Rotate Left**
-  **Rotate Right**
-  **Flip Vertical**
-  **Flip Horizontal**

The overlay is repositioned.

To Change the Display of an Overlay

1. To display the overlay at 50 percent opacity, click **Overlays** ▸ **Transparent**. The opacity is adjusted.
2. To display a different color where the geometries of the overlay and the design overlap, click **Home** ▸  **Overlap Color**. The overlap color is applied.

To Rename an Overlay

1. Click **Overlays** ▸  **Rename**. The **Rename Overlay** dialog box opens.
2. In the **Name** box, type a new name.
3. Click **OK**.

To Delete an Overlay

Click **Overlays** ▸  **Delete**.

To Set the Format, Scale, and Position Options for Gerber RS-274D Files

1. Click **Overlays** ▸ **Edit Options** ▸ **Gerber RS-274D Options**. The **Creo View Options** dialog box opens to the **PCB Layout Viewer-Gerber RS–274D** page.
2. Set options for these categories:
   - **Format**
   - **Units**
   - **Image Polarity**
   - **Scale**
3. To use a parameters file to set the options, under **Gerber Parameters File**, click **Load**. The **Browse Gerber Aperture File** dialog box opens.
4. Select the file and click **Open**.
5. Click **Apply** to apply the settings and keep the **Creo View Options** dialog box open, or click **OK** to apply the settings and close the dialog box.

Options for Gerber Files

Use these settings to control the parameters of Gerber RS-274D overlays:

Category	Option	Description
Format	**Integer Places** **Decimal Places**	Shows the selected number of digits for data coordinates. You can set the number of digits before and after the decimal point. For the RS–274D file, set these to 5 and 4, respectively.
	Leading zeros **Trailing zeros**	Drops the zero at the beginning or end of a data coordinate.
	Absolute	Displays data coordinates relative to 0.0.
	Incremental	Displays data coordinates relative to the selected coordinates.
Units	**Inches**	Sets the unit of

Category	Option	Description
	Millimeters	measurement.
Image Polarity	**Positive** **Negative**	Displays positive or negative images.
Scale	**Scale**	Sets the scale for coordinates.
Gerber Aperture File	**Browse**	Loads the file containing a description of geometry that defines the shape of the features.
Gerber Parameters File	**Load**	Loads a file containing settings for the parameters described in this table.

RS-274X files include definitions for these parameters.

To Set Scale and Position Defaults for Overlays

1. Click **Overlays** ▸ **Edit Options** ▸ **Overlay Options**. The **Creo View Options** dialog box opens to the **PCB Layout Viewer-Overlays** page.
2. To convert the scale for an overlay with a different unit of measurement, type a value in the **Scale** box.
3. To rotate the overlay, type or select an angle in the **Rotation** box.
4. To align the overlay origin to the design origin, type values in the **X Offset** and **Y Offset** boxes.
5. To flip the overlay, select one or both of these check boxes:
   - **On horizontal axis**
   - **On vertical axis**
6. Click **OK**. The defaults are set.

9

Searching the Structure

About Searching Structures and Designs

You can search MCAD structures for parts designs for object names or attributes. Enter the search text and conditions in one of the search dialog boxes. The Search function is context sensitive, so that MCAD data opens the MCAD search options, and an ECAD viewable opens the search tool to ECAD options. Search results appear by default in the  Results pane, but you can redirect them to be saved as a group.

You can narrow your search by using wildcard and codes to find words or phrases that contain specific letters or combinations of letters.

Types of Searches

The Creo View **Search** button is a split button. Click  **Search** on the icon to open the default **Search Product Structure** dialog box. The  **Find** functionality is also available in Creo View Lite. Click the arrow to select any one of the search options:

Search Types in Creo View

Type	Description
 **Search**	Narrows the search by defining several context sensitive search criteria to obtain a short, accurate results list. Searches the MCAD structure for parts or attributes, or perform a spatial search. Searches the ECAD design for objects or attributes.
 **Color-coded Search**	Performs multiple MCAD searches simultaneously and color-code the results.
 **Quick Search**	Obtains a list of entities by name. Searches by MCAD part name or ECAD object name.
 **Find**	Finds an entity with a known name, but unknown location.

Type	Description
	Searches by MCAD part name, attribute, or construction geometry entity, or ECAD part name or attribute.

MCAD Searches

About MCAD Searches

There are three types of MCAD searches:

Type	Searches by
Part	Name of entities
Attribute	Defined numeric or nominal attributes
Spatial	Parts within a physical distance from the bounding box or sphere center

The **Search Product Structure** dialog box updates dynamically based on the option you select.

Although you can run a search without giving it a **Query Name**, use the **Query Name** for these tasks:

- Upload the same search criteria in subsequent sessions.
- Enable multiple search criteria when applying the  **Color-coded Search** tool.

You can save the search results as a  **Search** group.

To Search for MCAD Parts

1. Open an MCAD structure.
2. Click **Home** ▸  **Search**. The **Search Product Structure** dialog box opens.
3. In the **Query Name** box, type or select a name for the search.
4. Choose a **Search Against** option from the list.
5. Next to **Do query of type**, select **Part**. The dialog box updates.
6. In the **Name** box, enter the text string you want to match.
7. Select a **Loaded in active view** option:

- **Don't Care**—Search all parts in the assembly, regardless of their load state.
- **Only Loaded**—Search only loaded parts.
- **Only Unloaded**—Search only on parts that are not loaded.

8. Select **Ignore Case** to disregard capitalization.
9. To define multiple search conditions at once, select the **Show Multi-line** check box, and then specify a combination of search rules to use in conjunction with each other. Select **And** or **Or** to define the conditions of the search.
10. Click **Apply** to run the search and save the Query Name. The search results appear under **Part Name** in the  Results pane in the upper data panel.
11. Click **Reset** to reset the query to when it was opened or when **Apply** was last pressed.
12. Click **Save As Group** to save the search results as a group.
13. Click **OK**. The search results appear in the **Results** pane in the upper data panel and the **Search Product Structure** dialog box closes.

To Group Search Results

1. Set the **Search** configuration in the **Search Product Structure** dialog box.
2. Click **Create Group**. The **Group Name** dialog box opens.
3. Type a name for the group click **OK**. The results appear as group members in the  Group pane.

To Search for MCAD Attributes

1. Open an MCAD structure.
2. Click **Home** ▸  **Search**. The **Search Product Structure** dialog box opens.
3. In the **Query Name** box, type or select a name for the search.
4. Select a **Search Against** option from the list.
5. In **Do query of type**, select **Attribute**. The dialog box updates.
6. Set the attribute variables as follows:
   a. Select a **Name** from the list or type a new name for the attribute. The **Name** list contains a list of all the different attribute names that exist in the current product structure.
   b. Under the **Categories to search**:
      - Select **Selected** to narrow the choice of property sets.
      - Select **All** to search all property sets.

Note

For one component, a single property often has multiple values.

c. Define the type of comparison:

- Check **Numeric Comparison** to apply the basic arithmetic operators list, or leave it unchecked to apply two string comparison operators list.
- Set the **Which is** variables.

  The **Which is** arithmetic operators are <, <=, =, !=, >, >=; and the two string comparison operators are `like` and `unlike`.
- Check **Value** and type a value in the adjoining box, or check **Attribute** and select an attribute name from the list. The search tool will then match the property against the named property on the same component.

Note

*If you use an arithmetic operator, a numeric comparison is performed, and if you use a string operator, a string comparison is performed. As a result, "prop name a" = "prop * b" will be converted into numbers for matching.*

7. Select **Ignore Case** to disregard capitalization.
8. To define multiple search conditions at once, select the **Show Multi-line** check box, and then specify a combination of search rules to use in conjunction with each other. Select **And** or **Or** to define the conditions of the search.
9. Click **Apply** to run the search and save the Query Name. The search results appear in the **Results** pane in the upper data panel.
10. Click **Save As Group** to save the search results as a group.
11. Click **OK**. The search results appear in the **Results** pane in the upper data panel and the **Search Product Structure** dialog box closes.

To Search an MCAD Structure Using Spatial Criteria

1. Open an MCAD structure.
2. Click **Home** ▸  **Search**. The **Search Product Structure** dialog box opens.
3. In the **Query Name** box, type or select a name for the search.
4. Select a **Search Against** option from the list.
5. In **Do query of type**, select **Spatial**. The dialog box updates.
6. Set the center of the spatial query by selecting a **Local to** option. The dialog box updates.
7. Continue as follows:

- **Sphere**
  a. Type the **X**, **Y**, and **Z** global coordinate values of the center of the sphere, or drag the sphere into position.
  b. Type a **Radius** value.
  c. Click the pin icon to the right of the coordinate values to lock the specified values. Click again to unlock the values.
  d. Check **Preview** to see the coordinate and radius positions.
  e. Drag the sphere's surface to resize it.
  f. Drag the green, red, or blue arrows to change the sphere's position.
- **Box**—Enter the **Minimum** and **Maximum X**, **Y**, and **Z** coordinate values to find the parts that fall within the bounding box.

  Note

  *Check **Preview** to see the coordinate and radius positions.*
- **Part Bounding Box**
  a. Select a **Part** from the Structure Tree or the graphics area.
  b. Click the pin icon to the right of the part name to lock the specified value. Click again to unlock the value.
  c. Enter the **Proximity** value to expand or shrink the bounding box in all directions. For example, a proximity value of 9 would expand the bounding box 9 units to the left, 9 units to the right, 9 to the top, and so on.

8. To define multiple search conditions at once, select the **Show Multi-line** check box, and then specify a combination of search rules to use in conjunction with each other. Select **And** or **Or** to define the conditions of the search.
9. Click **OK**. The search results appear in the **Results** tab in the Navigator.

About Color-coding Search Results

Use the  **Color-coded Search** tool to assign a different color the results of different MCAD search queries and view them together in order to differentiate the results on the Structure Tree and on the model in the graphics area. You can assign an extra color to accommodate search results that include items from more than one color category. Components that are not included in the search results are rendered semitransparent.

To Search by Color

1. Open an MCAD model.
2. Click **Tools** ▸  **Color-coded Search**. The **Color-coded Search** dialog box opens.
3. In the **Query name** box, type or select a name for the search.

   Note

   *This **Query name** is saved in a different folder to **Search** ▸ **Query name** items.*
4. Under **Assign colors to queries**, click **Add Condition**. A condition line appears in the box.
5. Fill in the condition line:
   - Under **Color**, click the button. The **Manage Colors** box opens. Select a color.
   - Check **Condition Type** ▸ **Named**. The **Search Name** list is populated with **Search** ▸ **Query name** items.
   - Select an item from the **Search Name** list to attach defined search criteria to the query.
   - To define new search criteria, click **Refine**. The **Search** ▸ **Search Product Structure** dialog box opens.
6. Under **Assign colors to queries**, click **Add Condition** to add another condition line to the box.
7. Click **Show matched multiple search conditions in**. The **Manage Colors** box opens. Select a color.
8. Click **Ok**. The search results appear in the **Results** tab of the Navigator.

Using Wildcards to Retrieve Part Names

You can enter wildcard characters to find part names. You cannot use wildcards in place of version numbers.

This is the match syntax:

- An asterisk (*) can replace any number of characters.
- A question mark (?) can replace any one character.
- You can type a question mark alone to get a complete listing of the appropriate names in the directory

For example, when you type : `*123?` you get the parts whose names match:

`<any characters>123<one character>` .

About Editing or Deleting Query Names

The **Manage Queries** button on the **Search Product Structure**, the **Color-coded Search**, and the **Search ECAD Design** dialog boxes opens the MCAD and the ECAD **Manage Queries** dialog boxes.

The **Manage Queries** dialog box lists all the saved queries by name. You can rename or delete queries from the list.

The MCAD **Manage Queries** dialog box includes the **Color-coded Search** option, allowing the user to choose either simple **Search** query names and **Color-coded Search** query names.

To Edit Search Query Names

1. With the MCAD **Search Product Structure** or the ECAD **Search ECAD Design** dialog box open, click **Manage Queries** . The **Manage Queries** dialog box opens.
2. Select an item from the list and click **Rename**. The **Rename Query** dialog box opens.
3. Type a new name and click **OK**. The name updates in the **Manage Queries** dialog box.

To Delete Search Query Names

1. With the MCAD **Search Product Structure** or the ECAD **Search ECAD Design** dialog box open, click **Manage Queries**. The **Manage Queries** dialog box opens.
2. Select an item from the list and click **Delete**. The name in the **Manage Queries** dialog box is deleted.

MCAD Search Results

About Working With MCAD Search Results

The  **Search** results open in the  **Results** pane in the upper data panel. Select items in the **Results** pane to view their corresponding entities on the Structure Tree and in the graphics area. The **Results** tab in the Ribbon contains these options:

- **Select in Structure**—Highlights the part currently selected in the Results pane in the graphics area and selects it in the Structure Tree.
-  **Color Code All**—Displays results in their respective colors in the Structure Tree and in the graphics area, making the correspondences between the product structure, assembly, and part in the query list easy to identify. Components that do not appear in the results list appear in semitransparent white in the graphics area.
-  **Clear Color Coding**—Removes color coding.
-  **Select all of Color**—Colors all parts in the graphics area that match the color code of the currently selected part in the results pane.
-  **Clear Results**—Empties the results pane and undoes color coding.
- **Show Query**—Opens the **Search Product Structure** dialog box and displays the configuration of the currently displayed search results.
- **Show Paths**—Toggles the display of the full path name in the Results pane.
- **Filter Results**

You can use the shortcut menu to activate the above-listed options.

To Filter Search Results

1. In the upper data panel, click  . The Results pane opens.
2. Right-click in the Results pane and choose **Filter Results**, or click **Filter** at the bottom of the results pane. The **Filter Results** dialog box opens.
3. To filter the items by name, check **Names matching** and type text in the **Enter text to match** field.
4. To filter results by color when the results of an Advanced Search are displayed, check **Colors matching**, and select one or more of the colors to display the items with the color.
5. Click **OK**. The results are filtered.
6. Right-click in the Results pane and choose **Reset Filters** to undo the filter.

ECAD Searches

About ECAD Searches

There are two types of ECAD searches:

Type	Searches by
Name or Object ID	Name of object or part
Attributes	Defined numeric or nominal attributes

The **Search Product Structure** dialog box updates dynamically based on which option you select.

Although you can run a search without giving it a **Query Name**, use the **Query Name** to upload the same search criteria in subsequent sessions.

To Search for an ECAD Object

1. Open an ECAD PCB Layout or Schematic file.
2. Click **Home** ▸  **Search**. The **Search ECAD Design** dialog box opens.
3. In the **Query Name** box, type a name, or select a name from the list.
4. In the **Search For** box, type a name, or select a name from the list.
5. Select an entity from the **Search Within** list, or leave the control at **All** the default value.
6. Select an option from the **Search By** list.
7. Select an option from the **Rule** list.
8. In the **Value** box, enter the text string you want to match when searching parts.
9. Select **Ignore Case** to query all text case options when performing the search.
10. Select **Show Multi-line** to enable the option, and then specify a combination of search rules to use in conjunction with each other. Select **And** or **Or** to define the conditions of the query. This option is unchecked by default and is relevant only if the query definition consists of more than one line.
11. Select **Apply** to run the search and save the Query Name. The search results appear in the **Results** pane in the upper data panel.
12. Select **Reset** to reset the query to when it was opened or when **Apply** was last pressed.
13. Click **OK**. If the query is named, it is saved. The query results appear in the  results tab in the Secondary Navigation pane.

To Search for ECAD Attributes

1. Open an ECAD PCB Layout or Schematic file.
2. Click **Home** ▸  **Search**. The **Search ECAD Design** dialog box opens.
3. In the **Query Name** box, type a name, or select a name from the list.

4. Select a **Search Within** category.

   Note

   *If you select **Selected** or **Highlight**, the **Search By** option automatically set to **Attribute**.*

5. Under **Search By**, select **Attribute**. The dialog box updates.
6. Select the **Value** or the **Name** radio button.
7. If you checked **Value**, then continue as follows:

   a. Select **All** or another option in the **Within Attribute** field which contains a list of all the different attribute names that exist in the current product structure.

   b. Select **Numeric Comparison** to apply the basic arithmetic operators list, or leave it unchecked to apply two string comparison operators list.

   c. Set the **Comparison** variables. The **Comparison** arithmetic operators are <, <=, =, !=, >, >=; and the two string comparison operators are `Equals` and `Does not equal`.

   d. Type a value in the **Value** box.

   If you selected **Name**, then continue as follows:

   a. Set the **Comparison** variable to `Equals` or `Does not equal`.

   b. Select an option from the **Value** list.

8. Select **Ignore Case** to query all text case options when performing the search.
9. Select **Show Multi-line** to enable the option, and then specify a combination of search rules to use in conjunction with each other. Select **And** or **Or** to define the conditions of the query. This option is unchecked by default and is relevant only if the query definition consists of more than one line.
10. Click **Apply** to run the search and save the Query Name. The search results appear in the **Results** pane in the upper data panel.
11. Click **Reset** to reset the query to when it was opened or when **Apply** was last pressed.
12. Click **OK** to run the query. If the query is named, it is saved. The query results appear in the  results tab in the upper data panel.

ECAD Search Criteria

Search Within —Leave at default **All**, or select one object type.

These objects apply to both Schematics and PCB layout:

- Components
- Nets

- Pins
- Selected
- Highlighted

These objects appear only when searching a PCB, and only if they have a unique identifier, such as Name/Ref Des, Type, Attribute, or Object ID to distinguish one from the other:

- Test Points
- Vias
- Regions
- Mechanical Holes

Search By—Narrows the search within the selected **Search Within** field.

Rule-Narrows the search further by filtering options such as **Top** or **Bottom** side for components, or **Routed** or **Unrouted** for nets.

Value—Enter alphanumeric values including variables and wildcard characters. The field is blank by default.

Working With ECAD Search Results

The  **Search** results appear in **Results** pane of the upper data panel.

The results are sorted alphanumerically. You can control the order from the column header. If you search for **All**, and the results include different types of objects, you can click in the icon column header to sort the results by object type. Each type has a different icon.

- Right-click and choose **Select** or double-click a result in the **Results** pane to view its corresponding entities on the Model Tree and in the graphics area, and its details in the lower data panel.
- Right-click and choose **Select All** and then **Select** in the results pane. A list of details of all the search results opens in the lower data panel.
- Right click and choose **Show Query** to open the **Search ECAD Design** dialog box with the configuration from which the displayed results were attained.
- You can change the query configuration, but if you click **Reset** , the configuration reverts to what it was when you opened the dialog box.
- **Clear Results** clears the results from the Results pane.

The Quick Search Tool

About the Quick Search Tool

Use the  **Quick Search** tool for MCAD or ECAD to obtain a list of entities with a common string in their names. You can type a new search string, or select an existing search string from the drop-down list.

To Perform a Quick Search

1. Do one of the following:
   - Click **Home** ▸  **Quick Search**.
   - Click the  **Results** tab of the Navigator.

   The **Quick Search** box at the top of the **Results** pane in the upper data panel is activated.
2. In the **Quick Search** box, type the text that you want to search for, or select the text from the drop-down list, then press ENTER. The results list appears on the  **Results** pane in the upper data panel.

Viewing ECAD Quick Search Results

The order for ECAD object names is:

- Components
- Nets
- Pins
- Test Points
- Vias
- Route Segments
- Regions
- Mechanical Holes

The Find Tool

About the Find Tool

You can find the location of MCAD parts and ECAD objects, as well as attributes of either in the loaded assembly using the  **Find** tool. you can also use the tool to look for words and phrases in PDF documents.

In the **Find** dialog box, you can enter a search string in the **Find what** box, and specify whether to search for part names, attribute names, or construction geometry. You can also select whether to match the whole word when searching, and whether to match the case of the text you entered.

The results are selected in the structure tree, and highlighted in the upper data panel. You can view the results individually or all at once.

To Search using the Find Tool

1. Click **Home** ▸ **Search** ▸ **Find**. The **Find** dialog box opens.
2. In the **Find what** box, type the text you want to search for.
3. Select one of the following definitions for the element that you want to find:
   - When finding an MCAD object:
     - **Part and Attribute name**
     - **Part name only**
     - **Attribute name only**
     - **Construction geometry only**
   - When finding an ECAD object:
     - **Object and Attribute name**
     - **Object name**
     - **Attribute name**
4. To return only exact matches, select the **Match whole word only** check box.
5. To make the find case-sensitive, select the **Match case** check box.
6. Do one of the following:
   - To find each instance of a name, click **Find Next** or **Find Previous**.
   - To find all instances of a specific name at one time, click **Find All**.

   The matching object attributes appear in the Data Panel and are highlighted in the Graphics Area and in the Structure Tree.

10

Viewing the Attributes

About Viewing ECAD or MCAD Attributes

An attribute is descriptive information to a selectable ECAD object or MCAD part or subassembly in Creo View. The attributes of these items are derived from source data and vary according to company procedures and design software. These are some examples of attributes:

- Numerical data
- Manufacturer
- Device Type
- Location of a product in a warehouse
- Product serial number

You can view attributes of the associated separate window. Object ID's enable cross-product collaboration between Creo Parametric, Creo View, and native ECAD tools. You also can copy attribute data to applications such as Microsoft Excel or Word.

In Creo View Lite and in Creo View you can view the attributes of one item at a time. In Creo View you can also view a list of attributes of multiple items.

Example: Viewing Attribute Information

View the attribute information in the  Attribute pane on the lower data panel. The name of a single selected part, subassembly, or object is appended to the pane title. The names of multiple items appear in the left column of the Attributes list. The objects are highlighted in the graphics area and the parts and subassemblies in the Model View.

The first four columns (**NAME**, **OBJECT TYPE**, **NET**, **COMPONENT**) in an ECAD attributes list are permanent. You can filter and rearrange the order of the other columns. The attribute value fields that do not require values are shaded, otherwise, the value field can remain empty.

To expand the lower data panel to show all attributes, click  in the top-right corner of the lower data panel. The contents are resized.

To View ECAD or MCAD Attributes

Presentation and Action	Result
As a vertical list—Select one ECAD object or MCAD part or subassembly in the graphics area, a search results list, or	The item and its attributes are displayed as a vertical list in the  Attributes pane in the lower data panel. A

Presentation and Action	Result
in the primary panel.	subassembly is displayed as one item.
On one line—Right-click the Attributes pane and choose **Show Single Select as List**.	The item and its attributes are displayed on one line.
Multiple items—Select multiple objects, parts or subassemblies in the graphics area or in the primary panel.	The items are displayed with their attributes as a list in the  Attributes pane.
To move an attribute to the fifth column —Right-click in an ECAD attributes column header to the right of the fifth column and choose **Move to 5th Column**.	The selected column shifts to the fifth column, and the order of the other columns is maintained.

About Filtering the Display of Attributes

You can filter the display of attribute columns for an ECAD object or an MCAD part or subassembly. If you select an attribute name shared by two of more ECAD objects, all objects with identical names are displayed.

To Filter MCAD Attributes

1. In the Attributes pane, right-click and choose **Filter Attributes**. The **Filter Attributes** dialog box opens.
2. To select the attribute categories to display, perform one of these actions:
   - Select **Categories matching** and **Category Name** and choose the categories.
   - Select **Names matching** and in the **Enter text to match** field, type the name of a category.
3. Click **Make Default** to keep the settings as the default.
4. Click **OK**. The Attributes list is filtered.
5. To revert to the original default settings, right-click in the Attributes pane and choose **Reset Filters**.

To Filter ECAD Attributes

1. In the Attributes pane right-click and choose **Filter Attributes**. The **Filter Attributes** dialog box opens.
2. Select a filter mode for the attributes:

- **Show All**—View all attribute columns.
- **Match Names**—In the **Enter text to match** box, type the name of a category to add to the attribute display.
- **Filter by object type and attribute names**—Select an **ECAD Object Type** from the list, or use the arrows to move attribute names between the **Available Attributes** and **Selected Attributes** lists.

3. Click **Make Default** to save the settings.
4. Click **Reset to default** to return to the original default settings.
5. Click **OK**. The Attributes list is filtered.
6. To revert to the original default settings, right-click in the Attributes pane and choose **Reset Filters**.

Display of ECAD Attributes

In a schematic view, select an object and the following attribute columns are displayed:

Object	Attribute Column	Displayed by Default
Symbol	Reference Designator	Yes
	Function/Device Type	Yes
	Sheet	Yes
	Part Number	
	Part Name	
Pin	Number	Yes
	Net	Yes
	Function	Yes
	Sheet	Yes
Wire	Net Name	Yes
	Sheet	Yes
Miscellaneous	Object Name (or unique ID)	Yes
	Other Attributes	

In a PCB layout view, select an object and the following attribute columns are displayed:

Object	Attribute Column	Displayed by Default
Component	Reference Designator	Yes
	Package Name	Yes
	Device Type	Yes

Object	Attribute Column	Displayed by Default
	Part Number	
	Part Name	
	Location	Yes
	Layer	Yes
	Rotation	Yes
	Height	
	Object ID	
Net	Net Name	Yes
	Manhattan Length	
	Routing Status	Yes
	Pin Name	Yes
Route Segment	Net Name	Yes
	Layer	Yes
	Width	Yes
	Length	Yes
Test Point	Name	Yes
	Net Name	Yes
	Layer	Yes
	Location	Yes
	Padstack Name	Yes
	Parent type	
	Parent name	
	Object ID	
Region	Board Outline	
	Type	
	Usage	Yes
	Object ID	
Via Pad	Net Name	Yes
	Size	Yes
	Location	Yes
	Padstack Name	Yes
	Padstack Layer Range	Yes
	Padstack Drill Size	
	Via Object ID	
Pin Pad	Pin Name	Yes
	Pin #1	
	Net Name	Yes
	Pad Size	Yes
	Location	Yes

Object	Attribute Column	Displayed by Default
	Padstack Name	Yes
	Padstack Layer Range	Yes
	Padstack Drill Size	
Pin-Pin Logical connection	Net Name	Yes
	Manhattan Length	Yes
	Route Length	
	Routing Status	
	Pin Names	Yes
Mechanical hole	Drill Diameter	Yes
	Style	
	Location	Yes
	Plating Status	Yes
	Layer Range	
	Net Name	
	Object ID	

11

Using the Structure Tree

About Modifying the Structure Tree

You can modify the Structure Tree to facilitate finding a specific part or subassembly. Use these methods to change display of parts in the Structure Tree:

- Navigating parts
- Filtering
- Positioning

These methods only affect the Structure Tree display. They do not affect the view of the model.

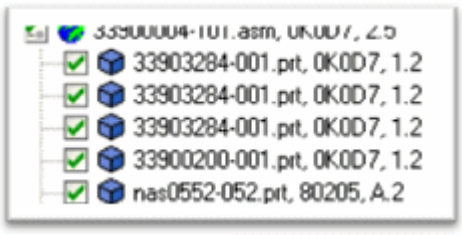
To Navigate the Structure Tree

Use the commands on the **Structure** pane to select nodes in the Structure Tree:

Action	Result
Click **Select Parent** ▸ **Select Parent**.	Selects parent of the current node.
Click **Select Parent** ▸ **Select Common Parent**.	Selects common parent of the selected nodes.
Click **Select Instances** ▸ **Select All Instances**.	Selects all instances of the current node.
Click **Select Instances** ▸ **Select Peer Instances**.	Selects all instances of the current node at the same hierarchical level.
Click **Select Children**.	Selects all descendants of the selected nodes.

About Node Names

The automated names of nodes in the Structure product design can be meaningless and confusing. You can use the **Configure Alternate Part Names** dialog box to configure more meaningful names based on the attributes of that part. For example, you can perform the following modification:

Default Structure Tree Display	Modified Structure Tree Display
	

To create alternate names for the parts you can select up to two attributes that can be preceded and followed by static text. You can save and reuse the settings in subsequent Creo View sessions.

To Configure Alternate Node Names

1. Open the **Creo View Options** dialog box in one of the following ways:
   - Click **File** ▸ **Creo View Options**.
   - On the **Structure** tab or the **Home** tab, in the **Structure Tree** group, click the arrow.
2. On the left, under **General**, select **Navigation**. The **General-Navigation** options open.
3. Select **Use alternate part names in the tree**, and then click **Configure**. The **Configure Alternate Part Names** dialog box opens.
4. In the first, third and fifth boxes enter text to be displayed. In the second and the fourth boxes, select attributes from the lists. These attributes are replaced by their value for the part being named. For instance, if you select material, the display shows steel or aluminum.

   Note

   - *The contents of the lists depends on the attributes in the PVS file.*
   - *The filter for the Structure Tree only works on the original names, not the alternate part names.*
   - *If a part has neither of the attributes selected, the original part name remains.*
5. Click **OK**. The **Configure Alternate Part Names** and the **Creo View Options** dialog boxes close, and the Structure Tree displays the new node names.
6. To revert to the original node names, open the **Creo View Options** dialog box, clear the **Use alternate part names in the tree** check box, and then click **OK**.

About the Structure Tree Filter

When you apply the Structure Tree filter, only nodes defined in the **Filter Viewables** dialog box are displayed. You can expand and collapse the nodes to see all or only the filtered nodes for a part or subassembly. The number of nodes viewed versus the total number of nodes is displayed. You can save the current filter settings for use across multiple sessions.

To Filter the Structure Tree Display

1. At the bottom of the Structure Tree click **Filter** or **Filtering**. The **Filter Viewables** dialog box opens.
2. Select **Part names matching** and type a string to display matching nodes and their ancestors.
3. Select **Only selected parts** to include only the parts selected and their ancestors.

   Note

   You can designate a maximum number of parts to display. If more than the designated number of parts are selected, then only the root node is shown.
4. Select **Results from a search query** to include the contents of search results even if they would be hidden by the filter.
5. Select **Add siblings of filtered parts** to include the siblings of the filtered parts.
6. Select **Dynamically re-apply filter if changes occur** to update the Structure Tree automatically according to the filters and commands. Otherwise it updates only when you click the refresh icon.
7. Select **Expand tree to show filtered parts** to show all the filtered in nodes and their ancestors. This is the default state.
8. Click **(click to expand)** to display the filtered out nodes from that location.
9. Select **Make Default** to save the current filter settings across multiple sessions.
10. Click **OK**. The filters are applied.

12

Creating and Editing Structures

About Creating and Editing Structures

You can insert viewables from other files into the Structure Tree and change the position of nodes and their children in the Structure Tree. You can also rename or delete nodes and save substructures as independent files. Although you can use Structure Edit only in Creo View, you can view the modified structures in Creo View Lite.

Some structure editing operations can break the internal references used with annotation sets of open views, causing the loss of unsaved work. These operations are grouped in the  **Advanced Editing** subcategory. Before performing these operations, you are prompted to save your changes.

After you insert new parts or assemblies into a structure, you can change the location and the orientation or both of the new objects using the **Assemble** tool.

Saving Substructures

You can select a subassembly and save it as an independent structure. You can include settings and viewables that are linked to the source structure in the new substructure file. These settings and viewables can be added to the new file:

- **Attributes**—Saves attributes linked to the relevant structure nodes. Selected by default.
- **Structure to root**—Saves the full structure path from the selected or visible entities to the root of the structure tree. If deselected, the selected or visible nodes are saved as individual root nodes. Selected by default.
- **Children of selected**—Includes children of selected nodes in the new file. Selected by default.
- **Specified file types**—Displays a list of file types that are available to include in the substructure file.

To Save a Substructure

1. In the Structure Tree, select the parts to save, and then click **File** ▸ **Save As** ▸ **Save Selected As PVS** or **File** ▸ **Save As** ▸ **Save Selected As PVZ**. The **Save Selected As PVS: Settings** or the **Save Selected As PVZ: Settings** dialog box opens.
2. Select any of these settings to include in the new file:

- **Attributes**—Saves attributes linked to the relevant structure nodes. Selected by default.
- **Structure to root**—Saves the full structure path from the selected or visible entities to the root of the Structure Tree. If cleared, the selected or visible nodes are saved as individual root nodes. Selected by default.
- **Children of selected**—Includes children of selected nodes in the new file. Selected by default
- **Specified file types**—Displays a list of available file types to include in the substructure file. If you select **Specified file types**, then select file types in the **Included** column.

3. Click **Save**. The **Save Selected As PVS: Settings** or the **Save Selected As PVZ: Settings** dialog box closes and the **Save Copy As PVS File** or **Save Copy As PVZ File** dialog box opens.
4. Select a location for the file.
5. In the **File name** box, accept the default file name, or type a new one.
6. Click **Save**.

Adding Viewables to Structures

About Adding Viewables to Structures

You can add a part or a subassembly to the structure at selected locations, or you can insert it into the structure root and then merge and reposition it afterwards.

You can also attach associated viewables, such as a PDF or a 2D illustration file to the structure.

To Add a Viewable to the Root of the Structure

1. Click **Structure** ▸  **Add Viewable To Root**. The **Open File** dialog box opens.
2. Browse to a file and click **Open**. The viewable is added at the bottom of the structure.

To Insert a Structure

1. Click **Tools** ▸  **Edit Structure** or **Structure** ▸  **Edit Structure**.
2. Select the target node, and then click  **Insert Branch Link** or  **Insert Branch**. The **Open File** dialog box opens.

3. Browse to and select a PVS, PVZ, ED, EDZ, or ZIP file. The selected file becomes the last child of the target node.
4. Click **Close Edit**.

To Add a Viewable to a Structure

1. Click **Tools** ▸  **Edit Structure** or **Structure** ▸  **Edit Structure**.
2. Select a node and then click  **Create Child Node**. The **Create Part** dialog box opens.
3. Type a name for the new node, and click **OK**. An empty node is added.
4. Select the empty node, and click  **Add Viewable**. The **Open File** dialog box opens.
5. Browse to and select a file, and then click **Open**. The viewable is added to the model.
6. Click  **Close Edit**.

To Create a New Structure

1. Click **File** ▸  **New**. The **Create new PVS File** dialog box opens.
2. Type the name for the new file and click **OK**. A message appears asking if you want to save the structure.
3. Click **Save As *.pvs** or **Save As *.pvz**. The **Save As PVS File** or the **Save As PVZ File** dialog box opens.
4. Type a name for the new file and click **Save**. The new file and the **Structure Edit** tool open.
5. Add a structure to the file.

About Advanced Structure Editing

The structure editing actions that can break internal references are grouped in  **Advanced Editing** mode. You are prompted to save or discard your changes before performing any of these actions:

- Deleting nodes
- Merging branch links—To move nodes into the root structure.
- Dragging and cutting and pasting nodes
- Renaming nodes

When you edit using the Advanced Edit tool, keep the following guidelines in mind:

- You can drag any node. All children are dragged along.
- All dragged parts move together to the target location.
- You cannot drag any substructure onto its own children.
- You can drag to and from a branch link. This updates the branch and the current structure.
- The physical location of parts does not change.
- The way parts are instanced might change.
- You cannot delete the top-level node.

Guidelines for Using Advanced Structure Editing

When you cut and paste using the **Advanced Editing** mode, keep the following guidelines in mind:

- You cannot use cut and paste in multiple locations simultaneously.
- **Cut** marks the selected nodes and does not remove them. The removal and placement change occurs with the **Paste** command.
- The **Cut** selection is discarded in the following circumstances:
  - When you insert a structure.
  - When the node is deleted.
  - When you close the Structure Edit tool.

To Delete a Structure

1. Click **Structure** ▸  **Edit Structure**. The **Structure Edit** tab opens.
2. Click  **Advanced Editing**. A message to warn you that you can lose unsaved changes appears. Select **Save** or **Discard** to proceed.
3. Select a node from the Structure Tree or from the graphics area and click **Delete**. A message to confirm the deletion appears.
4. Click **Yes**. The node and its children are deleted.
5. Click **Close Edit**.

To Rename a Part

1. Click **Structure** ▸  **Edit Structure**. The **Structure Edit** tab opens.
2. Select one or more parts to rename, and then click  **Advanced Editing**. A message to warn you that you can lose unsaved changes appears. Select **Save** or **Discard** to proceed.
3. Click **Rename**. The **Rename Part(s)** dialog box opens.
4. Type a new name for the parts and click **OK**. The parts are renamed.

Note

You can only set one new name at a time. If you select two or more parts, they will be renamed with the same new name.

To Cut and Paste Nodes

You can select more than one part to move at once, but you can only set one position for all of the selected parts.

1. Click **Structure** ▸  **Edit Structure**. The **Structure Edit** tab opens.
2. Select one or more parts to relocate, and then click  **Advanced Editing**. A message to warn you that you can lose unsaved changes appears. Select **Save** or **Discard** to proceed.
3. Do one of the following operations:
   - Click **Cut**.
   - In the Structure Tree, right-click and choose **Cut** from the shortcut menu.
4. Select the location on the Structure Tree to insert the parts, and click **Paste**. The selected parts are relocated in the structure.

To Merge a Branch Link

1. Click **Structure** ▸  **Edit Structure**. The **Structure Edit** tab opens.
2. Select one or more parts to merge, and then click  **Advanced Editing**. A message to warn you that you can lose unsaved changes appears. Select **Save** or **Discard** to proceed.
3. Click  **Merge Branch Link**. The branch links are merged.

The Assemble Tool

About Assembling Entities

Use the **Assemble** tool to accurately change the location or the orientation or both of one or more parts, groups, or assemblies in a model. Select the entities to move, and then define an origin and a destination coordinate system. The Origin coordinate system is the reference point for the selected entities. The Destination coordinate system is the reference point for the entities' new locations. The selected entities move as one rigid body. Use the linear and rotational offsets to fine-tune the position of the moved entities.

The reference collector in the **Selection** group shows the number of selected entities by type. You can view a list of the selected entities in the **Assemble: Selection List** dialog box, and you can remove entities from the list.

You can use the **Structure Edit** tool to insert new parts, groups, or assemblies into a structure, and then you can use the **Assemble** tool to position them.

To Assemble Coordinate Systems

1. Click **Tools** ▸  **Assemble**. The **Assemble** tab opens.
2. Select an **Assemble** mode:
   -  **Mate**—Moves and rotates the selected object
   -  **Orient**—Rotates the selected object
   -  **Locate**—Translates the selected object
3. In the graphics area or the Structure parts, groups, subassemblies, or assemblies to move, and then click  **Add Selected**.
4. Define the  **Origin CSYS** coordinate system. The Origin coordinate system appears.
5. Define the  **Destination CSYS** coordinate system. The Destination coordinate system appears.
6. To preview the move, click  **Preview**. Click  **Preview** again to undo the preview.
7. To adjust the destination position of the origin coordinate system, type or select linear and rotational offsets in the **Offset** group.

8. To approve the operation and create another one, click  **Apply**, or to approve the operation and exit the **Assemble** tool, click **OK**.
9. Click **Home** ▸  **Capture Location** and choose an option to save the new location, or **Home** ▸  **Restore Location** and choose an option to return the parts to their original locations.

To View and Modify the List of Parts to Assemble

1. Click the arrow in **Selection** group. The **Assemble: Selection List** dialog box opens with a list of selected items.
2. Select an entity from the list and click **Remove**. The entity is deleted.
3. Click **Close**.

Defining the Coordinate Systems for Assemble

The  **Origin CSYS** and the  **Destination CSYS** options follow:

- **Parts**—Calculates the  **Origin CSYS** from the objects that will move in the operation.
- **Global**—Calculates the  **Origin CSYS** from global coordinates. (The selected input model coordinate system is the default.)
- **Select from Graphics Area**—Opens the **Select Origin CSYS** collector. Select a part or a suitable edge, face, or construction geometry to define the coordinate system.
- **Define**—Opens the **Construct Coordinate System** dialog box. Create the coordinate system.

The most recently used coordinate systems are added to the **Origin CSYS** list for later use. You can select one from the list.

13

Creating Views

Editing Appearance

About Modifying Appearance

You can modify the appearance of one or in the model view by adjusting the following attributes:

- Render
- Color
- Transparency

You can select parts to modify from the model view. You can use these appearance modification tools to show hidden parts, emphasize contrasts between parts, or set a systematic color code. Use the visibility options to show, hide, or isolate specific parts or layers of a structure. You can modify appearance in Creo View in standalone mode. Saved appearances are read-only in Creo View Lite or Creo View Express.

About Render Mode

Render mode determines the appearance of the display:

Options	Function Key	Result
Shaded	F9	Displays the part as a shaded solid.
Shaded with Edges	F10	Displays the part as a shaded solid with the lines of forward surfaces emphasized.
Hidden Line Removal	F11	Displays only the frame lines of the forward surfaces.
Wireframe	F12	Displays front and back frame lines equally.
Mesh		Displays a mesh frame.

If you select **Hidden Line Removal**, **Wireframe**, or **Mesh**, all parts of the viewed representation are affected. If you select **Shaded** or **Shaded with Edges**, you can choose to set different colors and degrees of transparency to different parts of the structure.

If you select **Shaded** or **Shaded with Edges**, you can select one of the following modes:

- **Global Render Mode**—Displays all parts of the viewed representation as shaded.
- **Mixed Render Mode**—Displays the parts of the viewed representation as classified by their appearance attributes. This is the default working mode.

Note

Global Render Mode *or* ***Mixed Render Mode*** *options do not modify the parts' attributes, only their display on the screen.*

To Set Render Mode

1. Open a Creo View structure.
2. Open the **Render Mode** list in one of the following ways:
   - On the **Home** tab, in the **Display** group, click **Render Mode**, then choose an options from the list.
   - Press a function key.
   - Right-click and choose **Render Mode**, then point to an option from the list.
3. If you selected **Shaded** or **Shaded with Edges**, open the **Render Mode** list again and click **Global Render Mode** or **Mixed Render Mode**.

To Change the Render Mode

1. On the **Home** tab, in the **Display** group, click **Render Mode**. The **Render Mode** menu opens.
2. Set the extent of the render mode:
   - **Global Render Mode**—Applies the render mode to all components of the model.
   - **Mixed Render Mode**—Applies the render mode to selected components.
3. Select a render mode:
   - **Shaded**
   - **Hidden Line Removal**
   - **Wireframe**
   - **Mesh**

To Display Parts in Transparent Mode

1. Open a structure.
2. Select the parts of the structure to make transparent.
3. Set **Render Mode** to **Shaded** or **Shaded with Edges**.
4. Open the **Appearance** tab on the **Part(s) Properties** dialog box in one of the following ways:

- Right-click and choose **Edit Properties**.
- Click **Structure** ▸  **Edit Properties**. The **Part(s) Properties** dialog box opens. Click the **Appearance** tab.
- Click **Tools** ▸ **Edit Structure**. The **Structure Edit** tool opens. Click **Edit Properties**. The **Part(s) Properties** dialog box opens. Click the **Appearance** tab.
- On the **Home** tab click the arrow in the **Location** group. The **Part(s) Properties** dialog box opens. Click the **Appearance** tab.

5. Select one of the **Display Style** options:
   - **Shaded**
   - **Transparent**
6. Set the Transparency level by moving the slider, by typing in the percentage value, or by adjusting the value in the box. When the Transparency value is 100%, the selected part is completely transparent, whereas at 0% it is fully shaded.

   Note

   - *If the selected part is set to **Shaded** mode and transparency is set, the **Display Style** indicator switches to **Transparent**.*
   - *After the transparency level is set for the part, it remains the default value.*
7. You can select one of the **Restore Display Style** options:
   - **Restore to Original**—Returns the display style to the original structure value.
   - **Restore to Saved**—Returns the display style to the last saved style in the current annotation set.

To Set Transparency

1. Select a part of the structure.
2. On the **Model Parts** tab in the **Appearance** group, there are two display styles:
   - **Shaded**
   - **Transparent**
3. Click the mode you require. The selected part is displayed accordingly.

Note

*This option is enabled only if **Render Mode** is set to **Shaded** or **Shaded with Edges**.*

Rules for Setting Render Mode

If you select **Shaded** or **Shaded with Edges**, choose one of the following options:

- **Global Render Mode**—Displays all parts of the viewed structure with the same color and transparency attributes.
- **Mixed Render Mode** (default)—Displays parts of the viewed structure with different colors and degrees of transparency.

If **Global Render Mode** is selected and transparency is set, the mode automatically switches to **Mixed Render Mode**.

When the **Global Render Mode** and **Mixed Render Mode** are both available, the last checked option becomes active.

To Set Visibility Options

Use the visibility options to show, hide, or isolate specific parts or layers of a structure.

- **Isolate**—Shows only the selected parts.
- **Unhide All** and **Show**—Shows all hidden parts in the view.
- **Hide**—Hides the parts that are selected in the view.

The visibility options appear in the following locations on the Ribbon:

- On the **Home** tab in the **Display** group.
- On the **Structure** tab in the **Parts** group.
- On the **Structure Edit** tab in the **Parts** group.

To Edit Color

1. In an open a structure, select one or more parts to edit.
2. Right-click and choose **Color** or click **Model Parts ▸ Color**. The **Choose Color** dialog box opens.
3. Select the color and click **OK**. The color of the selected part updates.

   Note

   *To select a basic color when the **Choose Custom Color** dialog box opens, click **More**, and then select from the **Choose Color** dialog box.*
4. To reset colors, open the **Appearance** tab on the **Part(s) Properties** dialog box and select **Restore Color** and then choose a restore option:

- **Restore to Original**—Returns the color to the original structure value.
- **Restore to Saved**—Returns the color to the last saved style in the current annotation set.

Methods to Open the Choose Color Dialog Box

Open the **Choose Color** dialog box in one of the following ways:

- Right-click and choose **Color**. The **Choose Color** dialog box opens.
- On the **Model Parts** tab in the **Appearance** group, click **Color**. The **Choose Color** dialog box opens.
- On the **Structure** tab, click **Edit Properties**. The **Part(s) Properties** dialog box opens on the **Appearance** tab. Click the color indicator. The **Choose Color** dialog box opens.
- Click **Tools** ▸ **Edit Structure**. The **Structure Edit** tool opens. Click **Edit Properties**. The **Part(s) Properties** dialog box opens on the **Appearance** tab. Click the color indicator. The **Choose Color** dialog box opens.
- Right-click and choose **Edit Properties**. The **Part(s) Properties** dialog box opens on the **Location** tab.
  1. Click the **Appearance** tab. **Display Style** and **Color** options open on the right of the dialog box.
  2. Click the color indicator. The **Choose Color** dialog box opens.
- On the **Home** tab from the **Location** group:
  1. Click the arrow in the **Location** group. The **Part Properties** dialog box opens on the **Location** tab.
  2. Select the **Appearance** tab. **Display Style** and **Color** options open on the right.
  3. Click the color indicator. The **Choose Color** dialog box opens.
- Click **Color** on the **Model Parts** tab in the **Appearance** group.

Moving Parts

About Relocating and Reorienting Parts

You can use **Translation** mode to drag selected parts in the graphics area, to communicate design intent, and reveal hidden geometry. Use **Rotation** mode to spin the selected parts around their center.

You can rotate or translate a part in any of these ways:

- Use the commands on the Ribbon to drag and spin selected parts.
- Type specific dimensions on the **Location** tab of the **Part(s) Properties** dialog box.
- Relocate and reorient objects with the **Mate** tool.

The **Smart Explode** option moves parts along a specific axis and coordinate system to quickly create exploded views for illustrations or for viewing internal elements. You can use **Preset Explode** only if you have previously saved top-level explode states.

You can save location changes both in standalone mode and in Windchill.

To Translate and Rotate Parts Using the Location Dialog Box

1. Choose one or more parts in the Structure Tree.
2. Open the **Location** dialog box in any of these ways:
   - Right-click and choose **Edit Properties**.
   - Click **Structure** ▸  **Edit Properties**.
   - Click **Tools** ▸  **Edit Structure** ▸  **Edit Properties**.
   - Click the arrow on the **Model Parts** tab in the **Appearance** group or on the **Home** tab in the **Location** group.
3. Select one of the **Relative to** options:
   - **Parent**—Places the selected parts according to internal coordinate systems. Default when the selected parts are from the same parent.
   - **Global**—Places the selected parts according to the coordinate system of the whole model.
   - **Pick from geometry**—Create a reference coordinate systems on an object in the view.

   Note
   - *If you only change the referenced coordinate system, the coordinate system values change, not the part's location.*
   - *In case of multiple parts with offsetting, changing **Relative to** zeros the offset and keeps the previous state.*
4. Set the values in the **Translation** boxes to change the part location.
5. Set the values in the **Rotation** boxes to revolve the part.
6. To control the scale of the selected parts, set the **Scale Factor**.

7. To invert parts and create mirrored structures, click **Invert Part(s)**.
8. To return selected parts to the original or to a previously saved location, select a **Restore Location** option:
   - **Restore to Original**
   - **Restore to Saved**
9. Click **Home** ▸  **Capture Location** to save the location change.
10. Click **Close** to exit the dialog box.

To Move Parts

1. Activate the Translation mode in any of the following ways:
   - Press `F2`.
   - Right-click and choose **Home** ▸ **Translation Mode** from the shortcut menu.
   - Click **Home** ▸  **Translation Mode** or **Model Parts** ▸  **Translation Mode**.
2. Translate parts in the following ways:
   - Drag the green, red, or blue arrows to move selected parts along the x-, y-, or z-axis, relative to your viewing orientation.
   - Drag the white box to move parts in the plane of the graphics window.
3. Activate **Rotation Mode** in any of the following ways:
   - Press F4.
   - Right-click and choose **Location** ▸ **Rotation Mode** from the shortcut menu.
   - Click **Home** ▸  **Rotation Mode** or **Model Parts** ▸  **Rotation Mode**.
4. Rotate parts in the following ways:
   - Drag the colored rings in the virtual track ball to rotate selected parts.
   - Drag the second outer sphere to rotate the part in the viewing plane.
   - Right-click and drag the part in the graphics area to rotate the whole structure.
5. Click **Home** ▸ **Capture Location** and choose an option to save the new location, or **Home** ▸  **Restore Location**and choose an option to return the parts to their original locations.

Exploded Views

About Exploded Views

This feature is available for Creo View only.

Smart Explode lets you explode parts, subassemblies, or groups to move them along a specified axis and coordinate system, from an anchor that you select in the 3D view. Smart explode moves each item in the Explode list in the specified direction until its bounding box is clear of any parts not in the Explode list. You can explode parts individually or coupled together. You can view elements that are obscured, or create an assembly or disassembly sequence for training purposes with Smart Explode.

Preset Explode works explode states saved in Creo Parametric, and allows you to select from those explode statues. If no explode states are available, this option is dimmed.

To Perform a Smart Explode

1. Click **Tools** ▸  **Smart Explode**. The **Smart Explode** tab opens.
2. Click **Add Selected** and select one or more parts. The selected parts are highlighted on the representation and appear in the Explode list.
3. Choose an explode type from the **Add Selected** list.
   - **Smart Add**—Automatically adds the selected parts as single or coupled.
   - **Add Single Parts**—Add parts to explode individually.
   - **Add Coupled**—Add parts to move together.
4. Set the following options under **Direction**:
   - **CSYS**—Sets the coordinate system.
   - **Axis**—Sets the axis along which to move parts.
   - **Bidirectional**—Move the parts in opposite directions along the selected axis.
5. In the **Options** group, set from the following parameters as required:
   - Set the explosion **Anchor** option:
     - **Model Center** (default)
     - **Global**
     - **Non-Exploding Parts**—Sets a center that is calculated based on non-exploding parts.

    - **Select from Graphics Area**—Sets a user-defined reference point on the structure.
  - **Explode clear of all parts**—Moves all exploding parts until their bounding boxes do not overlap.
  - **Preview**—Select to preview the explode.
6. To preview the explode, click  **Preview**. Click  **Preview** again to undo the preview.
7. To approve the operation and create another one, click **Apply**, or to approve the operation and exit the **Smart Explode** tool, click **OK**.
8. Click **Home** ▸  **Capture Location** and choose an option to save the new location, or **Home** ▸  **Restore Location** and choose an option to return the parts to their original locations.

Sectioning

About Sectioning

You can section a structure to display internal alters the display of the structure, but it does not affect the geometry of the structure. Place one or two sectioning planes to cut the structure. You can make the following kinds of sections:

- **Planar**—A cut made by one plane
- **Quarter Cut**—A cut made by two perpendicular planes

When you modify the section, the views are You can save the section view as part of an annotation set.

-
-

To Section in the 3D View

1. On the **Sectioning** tab, in the **Setup** group, click **View Section**. A sectioning plane is previewed on the structure in the graphics area.
2. Click one of the following buttons to define the section type:
   - **Planar**
   - **Quarter Cut**
3. Move the sectioning plane or planes into position.

4. Set any of the following options:
   - **Show 2D View**—Opens the sectioned structure in a new drawing view.
   - **Cap Section**—Gives the parts the appearance of a solid interior.
   - **Show Boundaries**—Displays the sectioning plane or planes.
5. To exclude one or more parts from the cut, do the following operations:
   a. In the **Intersection** group, clear the **Intersect with all** check box.
   b. In the graphics area, select one or more parts that you want to section and click **Add Selected**.
   c. Continue selecting parts and clicking **Add Selected** until you have selected all the parts that you want to section.

Manipulating Sectioning Planes

Manipulate sectioning planes in the following ways:

Action	Result
In the 3D View	
Select and drag the plane.	Moves the plane in the direction that is normal to the plane.
Press and hold CTRL + select and drag the plane.	Rotates the plane around its center point.
On the **Sectioning** tab, in the **Setup** group, click **Initial Section** and select **X-Axis**, **Y-Axis**, or **Z-Axis**.	Places the plane along the selected axis.
In the 2D View	
Click + or −.	Moves the plane in the direction that is normal to the plane.
Drag the slider.	
Press the UP or DOWN arrow keys.	
In the **Section Properties** dialog box	
Under **Section Plane Orientation**, click **Set by Reference** and select a reference in the graphics area.	Positions the plane at a reference.
Under **Section Plane Orientation**, select an axis of rotation and type an angle.	Rotates the plane.
Under **Section Plane Orientation**, click **Flip**.	Flips the section view.
Under **Intersecting Point Location**, click **Set by Reference** and select a reference in the graphics area.	Positions the plane at a reference point.

Action	Result
Under **Intersecting Point Location**, select a translation direction and type a distance, or click + or −.	Translates the plane.

To Section in the 2D View

1. On the **Sectioning** tab, in the **Setup** group, click **Show 2D View**. A 2D view of the structure opens.
2. Click the graphics area of the 2D view. The view becomes active.
3. Move the sectioning plane into position.

Sections for 2D Views

When you are in Sectioning mode and you click **Show 2D View**, the 2D section window is displayed in a new view in the graphics area. Click this button again to remove the 2D section from the view.

You can perform the following operations in the 2D section view:

- Zoom
- Pan
- Measure
- Mark up
- Drag

When you modify the section in the 2D view, it is simultaneously updated in the 3D view. Use the slider to move the section plane. You can also move a section plane using the up and down keyboard arrows.

To Set Sectioning Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Defaults**. The **Defaults** options open on the left.
3. On the left, under **Model**, select **Sectioning**. The **Model-Sectioning** options open on the right.
4. To set the level of precision for sectioning, select or clear the **Use high level section detailing** check box.
5. To set the default sectioning behavior, under **Behavior**, select or clear any of the following check boxes:

- **Hide construction geometry while sectioning**
- **Hide all parts while sectioning**
- **Cap Sections**
  - **Use single color for section cap**—Select a color.

6. To outline the cut edges in a single color, select the **Use single color for section lines** check box and select a color.
7. Click **Apply** to apply the changes and keep the dialog box open, or click **OK** to apply the changes and close the dialog box.

14

Working with Markups

Annotation Sets and Folders

About Annotation Sets

An annotation set is a group of saved markups When you save an annotation set, the file is saved with the structure or design file. You must save an external file in a Creo View format before you can add and save an annotation set. You can create annotation sets in Creo View when working both with MCAD structures and ECAD objects. When working in Creo View Lite with MCAD structures you can create annotation sets, but when working with ECAD objects you can only view annotation sets. Annotation sets are on the  Annotation Sets pane of the upper data panel. The pane has two display modes:

- List View—Displays thumbnail previews of the annotation sets.
- Tree View—Lists the names of the annotation sets and annotation folders. You must be in Tree View to create, view, edit, and delete annotation folders.

These markups are saved to an annotation set:

- Annotations
- Measurements
- Construction geometry (MCAD only)

Animate the transition between animation sets by setting a **Loading** option. You can merge MCAD annotation sets and views by applying an annotation set to a base view or annotation set. In Creo View ECAD, additional properties are saved to the annotation set:

- Color and Visibility settings
- Navigation settings
- Highlights
- Isolation
- Overlays
- Images

If you designate an ECAD annotation set it opens when you load the design.

About Annotation Folders

Annotation folders are shown in the tree view of the Annotation Sets navigator. When you select a folder, thumbnail previews of the annotation sets in the folder are displayed in the lower half of the Annotation Sets Tree. You must be in Tree View to work with annotation folders. You can perform the following operations on Annotation Folders:

- Create new folders and name them according to their context.
- Edit folder name and content.
- Organize and reorder folders by dragging and dropping them in the tree view.
- Nest folders.
- Load the annotation set in the viewer.

You cannot keep an empty folder in the Annotation Sets Tree. When you remove all the annotation sets from a folder, the folder is automatically deleted.

To Open an Annotation Set

Click  Annotation Sets, and then select an action:

- Double-click the annotation set.
- Drag the annotation set into the graphics area.
- Right-click the annotation set and choose **Open in New View** from the shortcut menu.
- Right-click the annotation set and choose **Replace View** from the shortcut menu, and then select a view name to replace with the annotation set view.

To Animate the Transition Between Annotation Sets

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**. On the right, the global options open.
3. On the left, under **General**, select **Loading**. The **General-Loading** options open.
4. Select the **Perform animated transitions between files** check box.
5. In the **Transition duration** box, type or select a transition time in seconds. Transitions between annotation sets are animated.

Viewing InterComm Bookmarks in Creo View

Bookmark (`*.imx`) files created in InterComm Expert are read-only annotation sets in Creo View ECAD. They include this data:

- URL association with an ECAD design
- Name
- Text
- Redlines
- Highlighting
- Isolation
- Visibility
- Pan and zoom status
- Fill
- Color
- Crosshairs

About Applying an Annotation Set

You can merge annotation sets and views by applying an annotation set to a base view or annotation set. The base view must be open and loaded in the graphics area to apply an annotation set. When you apply an annotation set, the base view remains the active view. All the content from the base view remains, including content from previously-applied annotation sets, though some content may be hidden. The view retains the title of the base view. Each annotation set can contain only one of each of these data sets:

- Animation
- Design Check tags

Note

The base animation or Design Check tags are overwritten if you apply another annotation set with the same type of data set.

To Apply an Annotation Set to Another View

1. Open the base view or annotation set and load it in the graphics area.
2. Click  Annotation Sets.
3. Right-click the annotation set to apply and choose **Apply to View**, and then select the base view.

To Save an Annotation Set

1. Click **File** ▸ **Save Annotation Set** or **File** ▸ **Save Annotation Set As**. The **New Annotation Set** dialog box opens.
2. Type information in one or more of the following boxes:
   - **Name**
   - **Author**
   - **Telephone**
   - **E-Mail**
   - **Comments**
3. Click **OK**.

Note

*Annotation sets are also saved when you click **File** ▸ **Save All**.*

To Create an Annotation Folder

1. Make sure you have created and saved at least one annotation set.
2. In the Annotation Sets pane, right-click and choose **Switch to Tree View** from the shortcut menu. The annotation sets are listed in a tree.
3. Select the first annotation set to be included in the folder, right-click, and choose **Create a Folder** from the shortcut menu. A new folder is created with a default name.
4. To rename the folder, right-click and choose **Rename**, and then type a new name and press ENTER.

To Filter the MCAD Annotation Set List

1. In an open representation, select the Annotation Sets pane.
2. On the Annotation Sets pane, right-click and choose **Filter Annotations**. The **Filter Model List** dialog box opens.
3. Select the **Names matching** check box.
4. In the **Enter text to match** box, type the name of the annotation set that you want to find.
5. To set the filter as a Creo View option, click **Make Default**.
6. Click **OK**. The matching annotation sets are displayed on the Annotation Sets pane.
7. To remove the filter, right-click the Annotation Sets pane and choose **Reset Filters**.

To Filter the ECAD Annotation Set List

1. In an open design, select the Annotation Sets pane.
2. On the Annotation Sets pane, right-click and choose **Filter Files**. The **Filter Ecad List** dialog box opens.
3. Select any or all of the following check boxes. Type the names of the annotation sets that you want to find in the corresponding **Enter text to match** box:
   - **Names matching**
   - **File names matching**
   - **Part names matching**
4. Select or clear the **Only landmarks attached to selected parts** check box.
5. To set the filter as a Creo View option, click **Make Default**.
6. Click **OK**. The matching annotation sets are displayed on the Annotation Sets pane.
7. To remove the filter, right-click the Annotation Sets pane and choose **Reset Filters**.

Annotations

About Annotations

You can view and create annotations for every type of MCAD and ECAD viewable. Use the annotations to document the design, for product definition purposes, or as part of collaborative reviews. You can perform the following operations on annotations that you add during a view session:

- Edit
- Move
- Delete
- Hide or unhide
- Save

Note

A saved annotation set is stored to an `*.etb` *file that is linked to the structure file. You can also save an annotation set and upload it to the server.*

Annotations appear in the graphics area, and a list of the annotations appears on the  **View Content** pane of the upper data panel. In Creo View ECAD, when you select the annotation in either location, it is selected in both places.

To Create an Annotation

1. On the **Markup** tab, in the **Annotations** group, click an annotation type and select a subtype. The pointer becomes an annotation pointer.
2. Place the annotation using the corresponding placement method. When you create an annotation that includes a note, the **Note** dialog box opens.
3. Type the note and select the **Font Size**, **Font Color**, and **Background Color**, and then click **OK**.

General Annotation Types for MCAD and ECAD

Create these types of annotation for the following views:

Viewable	Annotation	Description
Model Drawing Image Schematic PCB Layout	**Note** **Note with Leader**	Adds explanation or details.
	Reference Note **Reference Note with Leader**	Marks a reference datum.
	Leader Line	Points to a spot on the representation. You can select a point on the leader to insert a new point at that location to move the leader line from that point. Any leader line can be divided into segments, and a new leader line can be merged with an existing note. Leader line settings such as head, tail, width, style, and color are defined in **Edit Properties ▸ Leader** options.
	Shape	Draws the following shapes: • Rectangle • Shaded rectangle • Ellipse • Shaded ellipse

Viewable	Annotation	Description
		• Polygon • Shaded polygon
	Freehand	Draws a freehand shape.
	Stamp	Adds a stamp to the representation. You can define stamp settings using the **Creo View Options** dialog box.
Model Drawing Image	**Gtol**	Adds Geometric Dimensioning and Tolerancing (GDT) symbols. GDT is an industry-standard language used by detailers to describe how components are fitted together. You can drag the GDT box to move it in the graphics area.

Annotation Types for the Document View

The following types of annotations are available on the **Markup** tab for the document view:

Annotation	Description
Note	Add explanation or details. You can define text size, color, and background in **Creo View Options** ▸ **Defaults** ▸ **Annotations** ▸ **Text**.
Note with Leader	
Leader Line	Points to a spot on the document. You can select a point on the leader to insert a new point at that location to move the leader line from that point. Any leader line can be divided into segments, and a new leader line can be merged with an existing note.

Annotation	Description
	You can define leader line settings such as head, tail, width, style, and color in **Creo View Options** ▸ **Defaults** ▸ **Annotations** ▸ **Line Styles**.
Shape	Draws the following shapes. • Rectangle • Polygon • Ellipse • Filled Rectangle • Filled Polygon • Filled Ellipse
Freehand	Creates a freehand annotation.
Text Highlight	Mark up text.
Text Strikethrough	
Text Underline	
Stamp	Adds a stamp to the document. You can define stamp settings using **Stamp** options.

Annotation Types for the Illustration View

The following annotation types are available on the **Markup** tab for the illustration view:

Annotation	Description
Note	Adds explanation or details.
Line	Points to a spot on the representation. You can select a point on the leader to insert a new point at that location to move the leader line from that point. Any leader line can be divided into segments, and a new leader line can be merged with an existing note. Leader line settings such as head, tail, width, style, and color, are defined in **Edit Properties** ▸ **Leader** options.
Shape	Draws the following shapes:

Annotation	Description
	• Rectangle • Ellipse
Freehand	Draws a freehand shape.

Placing Annotations

Different annotation types use different placement methods, as follows:

Type	Placement in the Graphics Area
Note **Reference Note**	Select a placement point.
Note with Leader **Reference Note with Leader**	Select an endpoint for the leader and drag it to the point where you want to place the note.
Leader Line	Select a start point for the leader line, and then select the end point. Double-click to finish creating the line. When you create a leader line with jogs, click once to place each jog. ***Note*** *For leader lines with arrowheads, the arrow is placed at the line's end point.*
Shape	Quadrilaterals and ellipses—Click and drag the shape to the desired size. Freehand polygons—Click once to create each corner of the polygon and double-click to finish creating it.
Freehand	Click and drag to draw the shape.
Gtol	Select a placement point and set the parameters in the **Gtol** dialog box.

To Hide or Show an Annotation

In the graphics area or the  View Content pane of the upper data panel, right-click the annotation and choose **Hide** or **Unhide** from the shortcut menu. The annotation is hidden or shown.

To Edit an Annotation

1. Right-click the annotation and choose **Edit Properties** from the shortcut menu.
2. Make your changes and click **OK**.
3. To rename the annotation, right-click it and choose **Rename** from the shortcut menu.

To Add Annotation Stamps

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **Annotations**, select **Stamps**. The **Annotations-Stamps** options open on the right.
3. Click **Add**. The **Add Stamp** dialog box opens.
4. Select a stamp file and click **Open**. The stamp is added to the list.

To Delete an Annotation

In the graphics area or the  View Content pane of the upper data panel, right-click the annotation and choose  **Delete** from the shortcut menu. The annotation is deleted.

To Delete All Annotations

1. Make sure a view with at least one annotation is open.
2. In the graphics area, right-click and choose **Delete** ▸ **Delete All Annotations**.

To Set Annotation Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, select **Annotations**. The **Annotations-Main** options open on the right.
3. Under **Annotation Set Save Defaults**, type default the signature details in the following boxes:
   - **Name**
   - **Telephone**
   - **E-Mail**
4. To receive a warning when you load a not support saving annotation sets, select the **Warn if a data set for which an annotation set cannot be saved is loaded** check box.

5. To change the default annotation appearance, in the **Showing** box, select **Defaults**.
6. On the left, under **Annotations** select an item to change its appearance:
   - **Line Style**—Sets a shape for the head and the tail, and sets the width, style, and color of the line.
   - **Text**—Sets a font size and a color for the text and the text background.
   - **Shape**—Sets a color for the shape.

Measurements

To Set Measurement Options

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Defaults**. The defaults options open.
3. On the left, under **General**, select **Measurement**. The **General-Measurement** options open on the right.
4. To mark approximated measurements, under **Measurement Precision**, select the **Mark with ~ approximated measurement results** check box.
5. To show or hide units, under **Display**, select or clear the **Show units** check box.
6. Under **Decimal Places**, set the degree of accuracy for measurements:
   - **Length**
   - **Angle**
   - **Volume / Mass / Area**

To Change the Unit of Measurement

1. To open the **General-Units** page of the **Creo View Options** dialog box, perform one of these actions:
   - In the Selection and Display Bar, double-click the unit of measurement.
   - Use the Application Menu:
     a. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
     b. In the **Showing** box select **Defaults** or **Active View**.
     c. In the **Showing** box select **Defaults**.
     d. Select **Units**. The **General-Units** dialog box opens.

2. Select a unit in one or more of these boxes:
    - **Default MCAD length units**
    - **Default MCAD mass units**
    - **Default MCAD density units**
    - **Default ECAD length units**
3. Click **OK**. The unit is changed for the active view.

Selecting Measurement References

When you select elements to measure, you can use one of the following selection types:

- Smart—Selects details within a representation.
- Part—Selects a part or an annotation.

Use one of the following methods to select a detail:

- Choose **Previous** or **Next** from the shortcut menu.
- Hold down CTRL and select the detail.
- Right-click and choose **Select From List** from the shortcut menu, and then select an object from the list on one of the following tabs:
    - **Parts & Markup**
    - **Part Geometry**

MCAD Measurements

About MCAD Measurements

You can create measurements for 3D models and 2D drawings. Measure one or more components or the distance or angle between components. Measurement results are displayed in the graphics area.

The measurements together with the reference to the parts measured are saved when you save the measurements in an Annotation Set.

Types of Measurements

Create the following kinds of measurements in Creo View:

- Distance
- Diameter/Radius

- Angle
- Summary
- Part dimension (Creo View only)

Measurement Units

You can set the default unit of measurement for the structure as part of the measurement options. The default units appear in the lower right corner of the graphics area. When the data is converted using an adapter or import filter, you can set an option to read and save the units of the original file. The default unit is applied when units conflict or the original units are not supported.

Note

For an assembly structure, the units of all components are the units from the top-level assembly. For drawings, each drawing can have its own units. As a result, when switching between the model view and the drawing view of a structure, the units may change.

The display and the decimal places are defaults settings that can be changed locally for each view. Measurement precision is a global setting for all views.

Summary and Part Dimensions

Summary measurements and part dimensions are sets of measurement data for one or more parts in the structure.

Summary Measurements

Measurement **Summary** includes the following information, depending on the representation you are measuring:

- 2D drawing—Total length of two or more entities
- 3D model—Total length, area, perimeter, and volume data for one or more edges, surfaces, or parts. If a part has a material assigned, the summary also includes the part's mass, centroid, and envelope.

When you select entities for **Summary**, the information is displayed in the graphics area with a leader line from the summary information to each item that is measured.

Part Dimensions

Part Dimension is a measurement of the bounding box around one or more parts. You can set a **Part Dimension** measurement to include one of the following data sets:

- Length, width, and height
- Coordinates for each of the corners

To Create an MCAD Distance Measurement

1. Click **Markup** ▸  **Distance**. The **Measure Distance** dialog box opens.
2. In the graphics area, select the start point for the distance measurement.

   Note

   In a model representation, click once to select the part, and then click again to select an edge or surface. Click a third time to select a point on the edge or surface.
3. In the **Measure Distance** dialog box, click the second reference collector.
4. In the graphics area, select the end point for the measurement. A minimum distance measurement is created.

To Create a Projected Distance Measurement

1. In the **Measure Distance** dialog box, click ☑. The dialog box expands.
2. Click  . The measurement is projected on the global coordinate system by default.
3. To change the coordinate system, select the **Projection on** reference collector, and then select a new coordinate system.
4. To change the direction of the measurement, select or clear the **X**, **Y**, or **Z** check boxes. The measurement is created in the directions that are selected.
5. To project the measurement on a face or an edge, select the **Projection on** reference collector, and then click on the face or the edge. The measurement is projected on the face or the edge.
6. Click  to remove the absolute distance, or leave it selected to include both measurements in the annotation set.
7. Click **OK**.

To Create an Angle Measurement

1. Click **Markup** ▸  **Angle**. The **Measure Angle** dialog box opens.
2. In the graphics area, select the start point for the measurement.

   Note

   Click once to select a part, and then click again to select an edge or surface.

3. In the **Measure Angle** dialog box, select the second reference collector.
4. In the graphics area, select the end point for the measurement.
5. To flip the angle, select **Use supplementary angle**.
6. Click **OK**.

To Measure a Radius or Diameter

1. On the **Markup** tab, in the **Measurements** group, click **Diameter or Radius**. The **Measure Diameter** dialog box opens.
2. In the graphics area, select the circle to measure.
3. Click **Measure Radius** to measure the radius instead of the diameter.
4. Click **OK**.

To Create an MCAD Summary Measurement

1. Click **Markup** ▸  **Summary**. The **Measurement Summary** dialog box opens.
2. In the graphics area, select a part to measure.
3. To add another part to the measurement, in the **Measurement Summary** dialog box, click **Add**, and then select the part.
4. Click **OK**.

To Create a Part Dimension

You can create part dimensions in Creo View only.

1. Click **Markup** ▸  **Component**. The **Bounding Box** dialog box opens.
2. In the graphics area, select one or more parts to measure.
3. In the **Bounding Box** dialog box, under **Show**, select one of the following measurement types:
   - **Extents**—Bounding box dimensions in the X, Y, and Z directions
   - **Corner coordinates**—Coordinate positions for each corner of the bounding box.
4. Click **OK**.

To Display Original Units

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Defaults**. On the right, the defaults options open.

3. On the left under **General**, select **Units**. The **General-Units** options open on the right.
4. Select the **Use same units as in the original file (when present)** check box.
5. For a warning when for an unsupported unit, select the **Show alert message when original units are not present** check box.

ECAD Measurements

About the ECAD Measurements User Interface

In Creo View Lite you can only measure distance for PCB layout views, and in Creo View you can measure all these dimensions for PCB Layout views:

- **Distance**—Distance between two points or two object vertices. You can also measure between the centers of these objects:
  - Pads
  - Holes
  - Route centerlines

  You can limit your selections to a layer or an object.
- **Minimum Gap**—Dielectric span between the edges of two objects. In some cases, such as between two large planes or conductive figures or between two long routes, more than one result can exist. You can scroll through all results in the **Measure Minimum Gap** dialog box.
- **Conductive Path**—Minimum electrical path between the centers of two objects on the same net. After you select the first object, all objects not on the same net are unavailable. The resulting measurement path is shown using a white line in the graphics area.
- **Summary**—All dimensions of a single object

Measurement results are displayed in the measurement dialog box and in the graphics area. When you modify the references, the result is automatically updated in both locations. When you close the measurement dialog box, you can choose to keep a measurement or discard it. You can save your measurements in an annotation set.

To Create an ECAD Distance Measurement

1. Click **Markup** ▸ **Distance**. The **Measure Distance** dialog box opens.
2. Under **Options**, select one of these rules:

- **Same Layer**—Limits the selection of the second point to the first point's layer.
- **Free Select**—Selects a point without snapping to an object or layer.
- **Same Object**—Limits the selection of the second point to the same object on which the first point resides.

3. Click the **From** box, and then select a location for the start point of the measurement. It appears in the first reference collector.

   Note

   To select a different point on the same object, you can use the mouse wheel to scroll through the options in the reference collector.

4. Click the **To** box, and then select a location for the end point of the measurement. It appears in the second reference collector, and the measurement appears under **Results**.
5. To adjust the measurement, click a reference collector and then select a new endpoint. Under **Results**, the measurement updates automatically.
6. To accept the measurement, click **OK**.

To Create a Minimum Gap Measurement

You can select two points in the graphics area, and then click **Markup** ▸  **Minimum Gap** to create the measurement or perform the following steps:

1. Click **Markup** ▸  **Minimum Gap**. The **Measure Minimum Gap** dialog box opens.
2. Click the **From** reference collector. The collector is activated.
3. Select a point. It appears in the **From** reference collector.
4. In the **Measure Minimum Gap** dialog box, click the **To** reference collector, and then select the second point for the measurement. It appears in the reference collector. The results appear under **Results**.
5. To scroll through multiple results, click **Preview Next Instance**. The next result in the list is displayed.
6. To accept the measurement, click **OK**.

Calculating the Minimum Gap on Different Layers

You cannot calculate distance between geometries on different layers. To calculate the minimum gap measurement between objects that spanned on more than one layer such as Pins, Vias, Nets or Drill holes, calculate the minimum gap between geometries related to the same layer.

For example: to measure the minimum gap between two pins, the first spanned on layers 1-3 and the second on 2-4, measure the distance between geometries on layer 2 taken from both pins and then measure the distance for geometries on layer 3 and take the minimum. This calculation gives us the minimum gap.

To Create a Conductive Path Measurement

Tip

To aid your selection of the references for this measurement, first isolate the net by selecting it and click ***Home*** ▸  ***Isolate*** ▸ ***Isolate****.*

You can select two pin pads or vias on the net in the graphics area and then click **Markup** ▸  **Conductive Path** to create the measurement, or perform the following steps:

1. Click **Markup** ▸  **Conductive Path**. The **Measure Conductive Path** dialog box opens.
2. Select a point on a net. It appears in the **From** reference collector, and all objects not located on the same net are dimmed.
3. In the **Measure Conductive Path** dialog box, click the **To** reference collector, and then in the graphics area, select a pin pad or via on the same net. It appears in the **To** reference collector.
4. In the **Measure Conductive Path** dialog box, under **Results**, select the type of result to display in the graphics area:
   - **Rat's Nest Line**—Shortest distance between the centers of two objects on the same net
   - **dX, dY**—Difference in X and Y positions between the centers of two objects
   - **Manhattan**—Direct, rectilinear path between the centers of two objects
   - **Path Length**
5. To accept the measurement, click **OK**.

To Create an ECAD Summary Measurement

You can select the object and then click **Markup** ▸  **Summary** to create the measurement, or perform the following steps:

1. Click **Markup** ▸  **Summary**. The **Measurement Summary** dialog box opens.
2. In the graphics area, select an object. It appears in the **Object** box and the results appear under **Results**.
3. To accept the measurement, click **OK**.

ECAD Objects Eligible for Measurement

Select points on any of these objects as a measurement reference:

- Board outline
- Component
- Geometry associated with a net
- Pad
- Via
- Figure
- Region
- Hole
- Test point

You cannot use these objects as measurement references:

- Padstacks
- Rat’s Nest lines
- Reference designators
- Pin numbers
- Markups

15

Working with Materials

Defining Materials and Calculating Properties

About Assigning Materials and Calculating Properties

When a part's material is defined, its mass and other material-dependent properties are calculated automatically. The creator of the part can define the material in the CAD tool, or you can assign a material to a part in Creo View. Select a default material to have automatically assigned to parts without a defined material. Create a list of materials manually, or import an XML (`*.xml`) file that contains each material's name, density, and unit of measurement. Use the **Creo View Options** dialog box to add, edit, or delete materials.

Using the **Part(s) Properties** dialog box, you can view the following information and copy it to the clipboard:

- Mass properties
- Inertia tensor
- Center of mass

You can also create a summary measurement to display the properties.

To Calculate the Mass of a Part

1. Select one or more parts, right-click, and choose **Edit Properties** from the shortcut menu. The **Part(s) Properties** dialog box opens.
2. On the left, click **Material**. The materials information opens on the right.

   Note

   *If the part has a material assigned, its properties are calculated automatically. The results appear under **Component Summary**.*
3. To view the data for the inertia tensor and the center of mass, click **Inertia Tensor**. The **Moments of Inertia** dialog box opens.
4. To copy data to the clipboard, click  in the **Part(s) Properties** or **Moments of Inertia** dialog box.

To Assign a Material to One or More Parts

1. Select one or more parts, right-click, and choose **Edit Properties** from the shortcut menu. The **Part(s) Properties** dialog box opens.
2. Under **Component Material**, click **Set Material**. The **Set Material** dialog box opens.
3. In the **Density Units** box, select a unit.
4. Select the material from the list and click **OK**. The properties are calculated and the results appear under **Component Summary**.

To Import a List of Materials

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**.
3. On the left, under **Model**, select **Materials**. The **Model-Materials** options open.
4. Click **Import**. The **Import Material** dialog box opens.
5. Browse to the folder with the list of materials, and select the file.
6. Click **Load**.
7. Click **Yes** to overwrite the current list of materials. The list is imported.

To Export a List of Materials

1. With a list of materials open in the **Model-Materials** page of the **Creo View Options** dialog box, click **Export**. The **Export Material** dialog box opens.
2. Browse to a folder for the report on materials.
3. In the **File name** box, accept the default name or type a new one.
4. Click **Save**. The report is exported.

To Add, Edit, or Delete Materials

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**.
3. On the left, under **Model**, select **Materials**. The **Model-Materials** options open on the right.
4. Click **Add** or select a material and click **Edit**. A dialog box opens.
5. In the boxes, type or select values for the properties:

- **Material Name**
- **Density**
- **Density Units**

6. Click **OK**. The material is added or its properties are updated in the **Materials** list.
7. To delete a material, select it and click **Delete**.
8. Click **OK**. The **Delete Material** dialog box opens.
9. Click **OK**. The material is deleted from the list.

To Set the Default Material

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**.
3. On the left, under **Model**, select **Materials**. The **Model-Materials** options open on the right.
4. Under **Materials**, select a material, and then click **Set as Default**.

Generating Reports

About a Bill of Materials

In Creo View, you can generate a Bill of Materials (BOM) for an MCAD structure or an ECAD design. Use the  **Column Chooser** in the  **Bill Of Materials** tool to select the data to include in the report. In the **Column Chooser**, you can also set the order of the columns. Select the columns to appear by default in the **Creo View Options** dialog box.

When you generate a BOM, a  **Bill Of Materials** pane opens in the lower data panel. The information you select to include is displayed in this pane in columns. You can sort each column by clicking the column heading. When you select an item in the **Bill Of Materials** pane, it is automatically selected in the graphics area and vice versa.

About an MCAD Bill of Materials

You can generate an MCAD BOM for all or part of a structure. Use the **Column Chooser** to configure and automatically update a BOM before or after it is generated. You can have one BOM for each view of the structure. Select one of these types to generate:

- **Parts only**
- **Assemblies only**
- **Top-level components only**
- **All (flat)**
- **All (hierarchical)**

You can export the BOM as a CSV file (`*.csv`) or as a Web page (`*.htm` or `*.html`). You can also use the shortcut menu to copy one or more of the rows to the clipboard.

About an ECAD Bill of Materials

You can create an ECAD BOM for the components of a Schematic or PCB Layout design. Create one BOM for each design as a whole. You can use the **Column Chooser** to select data to include in the report after you generate the report.

To Generate an MCAD Bill of Materials

1. With a structure open, click **Tools** ▸  **Bill Of Materials**. The **Bill Of Materials** tab of the ribbon opens.
2. Select a **BOM Type** from the box.
3. Specify the inventory for the BOM in the **Selection** box.
4. To select the information to include in the BOM:
   a. Click  **Column Chooser**. The **BOM Column Chooser** dialog box opens.
   b. Under **Available Fields**, select a category of information:
      - **System Variables**
      - **Properties**
   c. To add one or more columns to the BOM, select them in the list under **Available Fields** and click **Add**. The columns are added to the **Displayed Fields** list.
   d. To remove one or more columns from the BOM, select them in the list under **Displayed Fields** and click **Remove**. The columns are removed from the **Displayed Fields** list.
   e. To reorder the columns, select a column in the **Displayed Fields** list, and then click  or  to change its position.
   f. Click **OK**. The **BOM Column Chooser** dialog box closes.
5. Click **Bill Of Materials** ▸  **Generate BOM**. The bill of materials is created.

To Set Bill of Materials Options for MCAD

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Global**. On the right, the global options open.
3. On the left, under **Model**, select **Bill Of Materials**. On the right, the **Model-Bill Of Materials** options open.
4. To add one or more columns to the list of defaults, select them in the **Available Fields** list, and then click **Add**. The columns are added to the **Displayed Fields** list.
5. To remove one or more columns from the list of defaults, select them in the list under **Displayed Fields** and click **Remove**. The columns are removed from the **Displayed Fields** list.
6. To reorder the columns, select a column in the **Displayed Fields** list, and then click  or  to change its position.
7. Click **OK**. The options are saved.

To Generate an ECAD Bill of Materials

1. Click **Tools** ▸  **Bill Of Materials**. The **Bill Of Materials** tab of the ribbon opens.
2. Click **Bill Of Materials** ▸  **Generate BOM**. The bill of materials is created.
3. To select the information to include in the BOM:
   a. Click  **Column Chooser**. The **ECAD BOM Column Chooser** dialog box opens.
   b. Under **Available Fields**, select a category of information:
      - **System Variables**
      - **Properties**
   c. To add one or more columns to the BOM, select them in the list under **Available Fields** and click **Add**. The columns are added to the **Displayed Fields** list.
   d. To remove one or more columns from the BOM, select them in the list under **Displayed Fields** and click **Remove**. The columns are removed from the **Displayed Fields** list.
   e. To reorder the columns, select a column in the **Displayed Fields** list, and then click  or  to change its position.
   f. Click **OK**. The **BOM Column Chooser** dialog box closes.

About a Design Data Report

In Creo View, you can create these types of ECAD design report:

- **Object Specific**—Includes all available information about each component of the selected type.
- **Design Statistics**—Provides information about the creation of the design along with the number of components of each type. Create a **Design Statistics** report for the design as a whole, a group of selected objects, or a selected subcircuit.

For a **Object Specific** report, you can use the  **Column Chooser** before or after you generate the report to select the data to include. In the **ECAD Report Column Chooser** dialog box, you can also set the order of the columns. Some columns are  locked and cannot be removed from the report using the **Column Chooser**.

When you generate a report, a  report pane opens in the lower data panel. The information you select to include is displayed in this pane in columns. Rows of pin information associated with components are not included on the report pane, but they appear in an exported report. You can sort fixed columns with single-value entries by clicking the column heading. When you select an item in the report pane, the graphics area, or the substructure pane of the primary panel, it is automatically selected in all three locations.

To Generate a Design Data Report

1. Click **Tools** ▸ **Design Data**. The **Design Data** tab of the ribbon opens.
2. In the **Report Type** box, select **Object Specific**.
3. In the **Selection** box, select the object type for the report.
4. Click **Design Data** ▸ **Generate Report**. The report is created.

To Generate a Design Statistics Report

1. Click **Tools** ▸ **Design Data**. The **Design Data** tab of the ribbon opens.
2. In the **Report Type** box, select **Design Statistics**.
3. In the **Selection** box, select the objects to include in the report:
   - **Entire design**
   - **Selected objects**
   - **Highlighted objects**
4. Click **Design Data** ▸ **Generate Report**. The report is created.

To Export a Bill of Materials or Design Data Report

1. In an open BOM or report, perform one of these actions:
   - Click **Bill Of Materials** ▸ **Export**. The **Export Bill of Materials** dialog box opens.
   - Click **Design Data** ▸ **Export**. The **Export Report** dialog box opens.
2. Browse to the folder where you want to save the report.
3. In the **File name** box, accept the default name or type a new one.
4. In the **Type** box, select a file format:
   - `CSV`
   - `Web page`
5. Click **Save**. The report is exported.

16

Animating Structures

About Animation

Use the Animation module of Creo View to coordinate the components of an animation sequence and play back an animation. You can animate Model Views only, and you must have an Animation license to create and edit animations. You can play, create, and manage your animation using the Keyframe Editor, the ribbon, and the shortcut menu. Animate these types of items:

- Parts
- Subassemblies
- Assemblies
- Annotations
- Measurements
- Section cuts

Access to the Animation functions depends on your edition of Creo View and whether you have an Animation license installed.

Creo View Configuration	Animation Functions
Creo View with an Animation license	Create, edit, and playback
Creo View without an Animation license Creo View Lite Creo View Express	Playback only

Preview the path of a moving item in an animation by displaying a line with a coordinate system marking the position of the item at each key. You can play animations and set playback options to control the speed, looping, range, and direction.

Example: Possible Uses of Animation

Use Animation for these purposes:

- Visualize your assembly's operation. If you have the concept, but have not defined the mechanics, you can drag bodies to different positions and take snapshots to create an animation.
- Create an assembly or disassembly sequence animation of your structure.
- Create a maintenance sequence, a short animation of steps to take to instruct a user in how to repair or build a product.

- Conduct a packaging study for the product.
- Present the product in a specific environment for marketing purposes.

About Tracks and Keys

A track describes the motion in an animation. A key is a milestone in time that signifies a change in the animation.

Tracks

There are three types of tracks:

- Container—Contains one or more container or item tracks.
- Item—Animates a part or camera. Each item has its own track.
- Property—Includes the keys that define the motion and properties of a part, subassembly, markup, section, or camera.

Keys

When you create or edit an animation, a new key is created each time you perform one of these actions on an object or content:

- Move
- Modify
- Create

Each frame of the animation sequence contains a key for each recorded item. The type of transition between keys is the **Interpolation Mode**. Select one of these kinds of transitions for each key to define the movement in your animation:

-  **Step**—Changes only at the key's position on the timeline.
-  **Linear**—Transitions between keys at a constant rate.
-  **Spline**—Transitions between keys at an increasing or decreasing rate.

To Play an Animation

Perform one of these actions:

- On the playback bar, click  .
- Click **Animation** ▸ 

To Set Playback Options

1. Click **Animation**, and then set the following options in the **Play** group:
   - Playback speed—Select a relative speed from the box.
   - **Loop**—Loop the animation continuously.
   - **Range**—Set the start and end points in one of these ways:
     - Click  **Range**, and then set a **Start** point and an **End** point in seconds.
     - Drag  and  in the Keyframe Editor
     - Type start and end times in the  boxes in the Track Tree.
   - Direction—Click  to play the animation in reverse.
2. To toggle the display of the playback bar in the view from show to hide, perform one of these actions:
   - Click **Animation** ▸  **Playback Controls**.
   - Set the option in the **Creo View Options** dialog box:
     a. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
     b. In the **Showing** box, select **Global**.
     c. On the left, under **Model**, select **Animation**. The **Model**—**Animation** options open.
     d. Under **Animation Tool**, select or clear the **Show in-view playback bar** box.

Creating Animations and Movies

About Creating, Saving, Importing, and Exporting Animations

You can create an animation by recording There are two automatic recording modes:

-  **Record Content**—Records changes to the position of items, markup, and sections.
-  **Record Camera**—Records changes to the camera's position.

In the automatic recording modes, a recording indicator  appears in the graphics area and the status bar. Tracks are added in the Keyframe Editor when you move a new item or create new content. Every time you make a change, the timeline in the Keyframe Editor is extended by an increment and a new key is created. The manual recording mode is called  **Capture**. Use it to take a manual snapshot of selected items, camera positions, or both. New tracks are created for cameras that do not already have them. In  **Record Content** mode, you can add these effects to one or more parts:

-  **Fade Out**—Adds a transition from opaque to transparent.
-  **Fade In**—Adds a transition from transparent to opaque.
-  **Screw**—Rotates the part.
-  **Unscrew**—Rotates the part.
-  **Color**—Adds a transition from the original color to another color.

You can import Creo Parametric (`*fra`), and you can open ProductView Standard animations. You cannot, however, open Creo View animations in ProductView Standard. The imported animation must match the current structure. An imported animation replaces the current animation. It may contain information about the conversion factors for time and space definitions, or you may have to specify them in the import dialog box. Animations are saved in an annotation set. You can also export an animation as a movie. Choose an encoding format from the list of codecs available on your computer.

To Create and Save an Animation

1. With a Model View open, click **Animation** ▸  **Advanced**. The Animation tool opens.
2. Click **Animation** ▸  **Animations** ▸ **New**. The Keyframe Editor opens.

   Note

   The first time you open the Animation tool for a view, a new animation is created automatically.
3. To set the recording mode, click **Animation** ▸  **Record Content** or **Animation** ▸  **Record Camera**.
4. For each part, subassembly, or assembly to move, perform these actions:
   a. Select the item to move.

b. Select a motion type.

c. Drag the item to a new location in one or more increments.

5. Save the animation as an annotation set.

To Add or Change an Animation Effect

1. In an open animation, select one or more parts.
2. Click the arrow below the current effect. The default effect is  **Fade Out**.
3. Select an effect from the list:
   -  **Fade Out**
   -  **Fade In**
   -  **Screw**
   -  **Unscrew**
   -  **Set Color**

To Preview the Path of an Item

1. In the Track Tree, select the item's track.
2. Click **Track Tools** ▸  **Preview**. The item's path appears. You can preview the path of only one item at a time.

You can also right-click the track and choose **Preview** from the shortcut menu.

To Import an Animation

1. Click **Animation** ▸  **Advanced**. The Animation tool opens.
2. Click  **Animations** ▸ **Import**. The **Import Animation** dialog box opens.
3. Click **Yes** to confirm the import. The **Load Animation** dialog box opens.
4. Select the animation file to import and click **Load**. The animation is loaded, or the **FRA file importer settings** dialog box opens.
5. Specify time and space information about the animation:
   a. Select a conversion factor in the boxes next to **Units**.
   b. In the **Frames per second** box, select or type a value.
   c. Click **Import**. The animation is loaded.

To Export an Animation as a Movie

1. Click **Animation** ▸  **Export**. The **Export Movie : Settings** dialog box opens.
2. In the **Save to file** box, type a path for the movie, or click **Browse** to browse to the file location.
3. In the **Record using codec** box, select an encoding format.
4. Select values for the following options:
   - **Frames per second**
   - **Width**
   - **Height**
5. To maintain the animation's proportions, select the **Lock aspect ratio** check box.
6. To play the movie automatically after creating it, select the **Launch movie player when complete** check box.
7. Click **Create movie**. The movie is created.

Editing Animations

About Editing Animations

In Creo View, you can edit an animation using these methods:

- Add or remove tracks or keys
- Modify tracks
  - Drag
  - Lock or unlock
  - Place in a container track
  - Reverse
  - Mirror
  - Merge
  - Split
  - Stretch
- Modify keys
  - Edit properties
  - Drag
  - Duplicate
  - Cut

- Copy
- Paste
- Change mode
- Delete

You can find these tools in the Keyframe Editor and on the shortcut menu. You can also access the tools from the **Track Tools** and **Key Tools** tabs of the Ribbon.

To Edit the Keys Using Different Methods

Access each of the key editing operations in one of these ways:

Operation	Keyframe Editor	Key Editor Dialog Box	Key Tools
Move	Drag the key to a new frame.	1. Right-click the key and choose **Edit** from the shortcut menu. The **Key Editor** dialog box opens. 2. Type a new value in the **Time** box.	1. Select the key. 2. Click **Key Tools**, and then select or type a value in the  **Time** box.
Change interpolation mode	Right-click the key and choose **Mode** from the shortcut menu, and then select a mode from the list: • **Step** • **Linear** • **Spline**	1. Right-click the key and choose **Edit** from the shortcut menu. The **Key Editor** dialog box opens. 2. In the **Interpolation Mode** box, select a mode: • **Step** • **Linear** • **Spline**	1. Select the key. 2. Click **Key Tools**, and then click the arrow under the current interpolation mode. 3. Select a new mode: •  **Step** •  **Linear** •  **Spline**

Operation	Keyframe Editor	Key Editor Dialog Box	Key Tools
Duplicate	1. Right-click the key and choose **Duplicate** from the shortcut menu. The key is duplicated and selected. 2. Drag the key to a new frame.		1. Select the key. 2. Click **Key Tools** ▸  **Duplicate**. The key is duplicated and selected. 3. Drag the key to a new frame.
Cut Copy Paste	Right-click the key and choose **Copy** from the shortcut menu, or right-click a frame and choose **Paste** from the shortcut menu.		1. Select a key or a frame. 2. Click **Key Tools**, and then click one of the following: •  **Copy** •  **Cut** •  **Paste**

Duplicating and Copying Keys

Use the **Duplicate** command to create an identical key that appears on top of the original. After you perform the **Duplicate** operation, the new key is automatically selected, and you can drag it to a new location. Use the **Copy** command to create a copy of the key on the clipboard. Then, paste it in a new location on the timeline, or paste it in another animation.

About Easing Movement

Easing is the acceleration or deceleration of a part or a camera's movement. You can add easing to any number of keys, and you can perform the operation on one or more keys at once. A key must have the **Spline** interpolation mode set to add easing. Each ease controls the velocity of the item at these points around the key:

- **Ease to**—Approach
- **Ease from**—Departure

The body moves according to the ease value you set:

- **Slow**—Decelerates
- **Normal**—Travels at a constant velocity
- **Fast**—Accelerates

For example, when you select a key at the 5–second mark and set the **Ease to** value to **Slow** and **Ease from** to **Normal**, the object decelerates as it approaches 00:05. After the 5–second mark, the object continues traveling at a constant speed toward the location set by the next key.

To Accelerate or Decelerate a Body Using Easing

1. Right-click a key and choose **Edit** from the shortcut menu. The **Key Editor** dialog box opens.
2. In the **Interpolation Mode** box, select **Spline**.
3. Click **Control**. The **Spline Control** dialog box opens.
4. Select a value in one or both of the boxes under **Transition**:
   - **Ease to**
   - **Ease from**
5. Click **OK**.

To Edit Tracks Using the Track Tools

Use the **Track Tools** to modify one or more tracks of the animation as follows:

Tool	Description
Lock	Locks the track to prevent unintended changes.
 **Focus**	Shows in the Keyframe Editor the selected tracks and hides all other tracks.
 **Insert Keyframe**	Adds a new key at the location of the time insertion point.
Start and **Duration**	Defines the exact start time and duration in seconds of a track.
 **Split**	Slices the track at a selected location.
 **Merge**	Splices together two consecutive segments on the same track. The segments do not have to be adjacent.

Tool	Description
Mirror	Creates a copy of the track whose keys are reversed relative to the original.
Reverse	Reverses the order of the keys in the track.

To Insert Tracks and Keys

1. In the Keyframe Editor, select the location for the insertion:
   - Key—Any point in the timeline
   - Track—Another track
2. Click **Track Tools** ▸ **Insert Keyframe** or **Track Tools** ▸ **Insert Folder**. The item is inserted.

You can also insert a track by dragging a part from the Structure Tree to the Keyframe Editor.

To Stretch a Track

1. In the Keyframe Editor, select the beginning or end of the track to stretch.
2. Drag the track end to a new location on the timeline. The track is stretched.

To Delete a Track or Key

In the Keyframe Editor, right-click the track or key and choose **Delete** from the shortcut menu.

You can also use these tools on the ribbon:

- **Track Tools** ▸ **Delete**
- **Key Tools** ▸ **Delete**

About Locking and Unlocking Tracks

You can lock one or more tracks in an animation. Locking prevents an unintentional change to the track, but it does not limit access to changing it. When you lock a parent track, its children are also locked.

To Lock or Unlock a Track

1. To lock a container or track, right-click it and choose **Lock** from the shortcut menu.
2. To unlock, right-click the container or track and choose **Unlock** from the shortcut menu.

   You can also select a track and then click **Track Tools** ▸ **Lock** to toggle its lock status.

17

Detecting Interferences

Interference Detection

About Interference Detection

Use Interference Detection to determine whether parts of a model overlap, touch, or are too close to each other. An interference detection test is called a query. You can save the query settings and the results. One user can define the query and another user can perform it. You can also compare different query settings to the same model input and examine different iterations of the same model using the same query settings.

You must have an Interference Detection license to use the Interference Detection tool.

Creo View Edition	Interference Detection Functions
Creo View with an Interference Detection module license	Read and write
Creo View without an Interference Detection module license	Read only View results and open reports.
Creo View Lite	Read only
Creo View Express	None

You can set Creo View to open Interference Detection information as read-only data.

To Perform an Interference Detection Query

1. Click **Tools** ▸  **Interference Detection**. The **Interference Detection** tab and the **Interference Detection Settings** dialog box open.
2. Select a **Query Name** from the list or type a new one.
3. Select an option from the **Calculation Mode** list.
4. Select an interference **Detect** option of the following type or combination of types:
   - **Interference**
   - **Contact + Interference**
   - **Clearance + Contact + Interference**
5. Type a value for one or more of the following boxes:

- **Tolerable Penetration**
- **Tolerance**
- **Required Clearance**

6. Create an input list in the **Input** panel.
7. To test interference between entities, create a second input list by clicking  . The second **Input** panel opens.
8. Create the second input list.
9. To test parts within the assembly or group, click the **Self Intersection** check box.
10. To exclude parts or assemblies from the tests, click the **Exclusion** tab and create an exclusion list.
11. Click  to leave the dialog box open after the calculation.
12. Perform the following steps to save the query settings:
    a. Click **Manage Queries**. The **Manage Queries** dialog box opens with the given **Query Name** inside.
    b. Click **Save**. The **Name** dialog box opens.
    c. Click **OK** to save the query, or change the name and then click **OK**. The query name is added to the list in the **Manage Queries** dialog box.
    d. Click **Close**. The **Manage Queries** dialog box closes.
13. Click **Calculate**. The results appear in a list in the  Interference Detection panel and graphically in the **Interference View**.

Calculation Mode Options

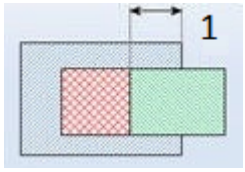
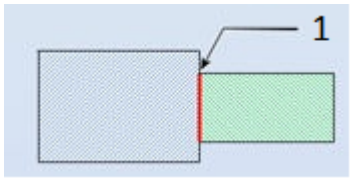
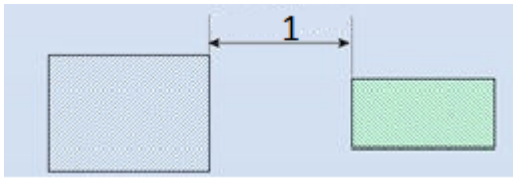
Calculation mode uses the following types of geometric data as options:

- **Quick**— Approximate geometry
- **Accurate**— B-rep geometry

Note

*If you select **Accurate** for models with mixed data, the **Accurate** option is run for parts with B-rep and the Quick option is run for other parts.*

Example: Interference Detection Options

Detect Option	Detection	Figure
Interference	Parts overlap in 3D	 1 Tolerable Penetration
Contact	Parts touch one another, but do not overlap	 1 Tolerance
Clearance	Parts are closer than the defined minimum distance	 1 Required Clearance

Setting Interference Detection Types

You can select one of these interference detection types or combinations:

- **Interference**
- **Contact** and **Interference**
- **Clearance**, **Contact**, and **Interference**

Interference Detection Type	Default Tolerance Value	Interference Detection Value
Interference	**Tolerable Penetration** = 0	Negative
Contact	**Tolerance** = 0	Positive
Clearance	**Required Clearance** = 1	Positive

Note

- *The tolerance value boxes match the interference detection type.*
- *The **Tolerable Penetration** value must be greater than or equal to 0. The box appears only when **Interference** is selected.*
- *The **+/-** symbol appears in the **Tolerance** value prefix, as the value is symmetrical.*
- *The **Required Clearance** value cannot be less than the **Tolerance** value.*

Input of Entities for Interference Detection

Use the following entities as input in an Interference Detection query.

- Parts
- Assemblies
-  Groups—Dynamic or Static

Input can be grouped in these ways:

- **Self Intersection**—Checks interferences inside assemblies and groups. The default state of **Self Intersection** for assemblies and groups is selected.
-  —Divides the input into two groups to test for interferences between them.

To Create Input Lists and Exclusion Lists

1. To create the **Input** list for interference detection, move items to the **Input** panel in one of the following ways:
    - Drag parts or assemblies from the Structure Tree.
    - Select one or more parts on the Structure Tree and click  Selected Part.
    - Drag groups from the Groups pane.
    - Click a shortcut commands on the **Input** tab:
        -  Structure—All entities in the Structure Tree
        -  Loaded—All loaded parts
        -  Selected—The selected group
        -  Visible—All loaded and visible entities
2. Create the **Exclusion** list in one of these ways:

- Drag parts or assemblies from the Structure Tree or groups from the Groups pane to the **Exclusion** panel.
- Right-click an item in the **Input** list, and then and choose **Exclude**.
- Right-click an item in the Results list, and then and choose **Exclude Pair**. Select the **Instances** check box to exclude all instances of the selected pair.

3. To remove an entity from the **Input** list, right-click and choose **Clear**.
4. Save the query settings as an annotation set to use them in future tests or to compare the results for future model iterations.

About Grouping Interference Detection Input

When working with large models, it is good practice to create groups to drag into the **Input** and the **Exclusion** lists for interference detection. You can organize groups in the following ways:

- Create Static groups by adding parts from different assemblies to the input list.
- Populate Dynamic groups automatically by attribute types.
- Define Dynamic groups by creating an assembly hierarchy.

After you perform interference detection, you can select a result and add it to the **Exclusion** list. In this way, you can exclude known and expected interferences, such as bolt-bracket pairs, from the Results list. You can also exclude all instances of the pair.

When you select a pair group to exclude, the prompt `(Not Synchronized)` appears next to the Annotation Set name in the graphics area. Recalculate the interference detection.

About Managing Queries for Interference Detection

The **Manage Queries** dialog box shows all available queries for the loaded file by name. If you create a query without naming it, you cannot save it. You can rename or delete queries from the list.

- The **Local** check box shows the queries that were saved per the specific loaded data set. By default, new queries are saved from the current data set as **Local** queries.
- The **Global** check box shows the queries that were set as global per any data set. Global queries do not include the input as input is data-specific.

To Set Local Queries to Shared Queries

1. In the **Interference Detection Settings** dialog box, click **Manage Queries**. The **Manage Queries** dialog box opens.
2. Right-click an item in the **Local** query list and choose **Set As Shared Query**, then click **Save**.

To Rename Interference Detection Queries

1. In the **Interference Detection Settings** dialog box, click **Manage Queries**. The **Manage Queries** dialog box opens.
2. Select an item from the list and click **Rename**. The **Rename Query** dialog box opens.
3. Type a new name and click **OK**. The name updates in the **Manage Queries** dialog box.

To Delete Interference Detection Queries

1. In the **Interference Detection Settings** dialog box, click **Manage Queries**. The **Manage Queries** dialog box opens.
2. Select an item from the list and click **Delete**. The query is deleted.

To Set Message Options for Interference Detection

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, click **Interference**. The **Model-Interference** options open.
3. Under **Messages**, select one or both of the following check boxes:
   - **Show information message when interference calculation is completed**
   - **Check for unsaved reports**
4. Click **Apply** to apply the options and keep the **Creo View Options** dialog box open, or click **OK** to apply the options and close the dialog box.

Working with Interference Detection Results

About Interference Detection Results

Save the interference query as an XML `*.pvc` annotation set file. The file includes both the interference settings and the calculation results. Each saved annotation set contains the results of only one calculation.

The results are generated as a list of pairs of entities between which interference is detected. The Results list appears in the  Interference Detection pane of the lower data panel, and an image of two interfering parts of a selected row appears in the **Interference View** in the graphics area.

You can add notes and predefined tags to each detected interference in the Results list to monitor Interference Detection results. You can create and export a **Summary Report** and a **Full Report**.

About Monitoring Interference Detection Results

Tags and notes can convey the information that a tag can convey:

- Status
- Related tasks
- Assignment

Your tags and notes remain in the Interference Detection results and become part of the **Full Report**. Custom tags and default tags can be modified and deleted. A box for optional legend is provided.

Four tags are included with Creo View:

Symbol	Highlight Name	Description
	Accept	Approves the object
	Reject	Rejects the object
	In Work	Flags the object for checking
	None	Adds the object without assigning a specific status

To Create, Reply to, and Delete a Note in Interference Detection Results

1. Select a result in the  Interference Detection pane.
2. Right-click and choose **Note** or click **Interference Detection** ▸ **Note** to open the **Note Properties** dialog box.
3. Type the note, and then click **OK**. A note box appears in the **Note** column of the selected result.
4. To open the note, right-click the result and choose **Note**. The **Note Properties** dialog box opens.
5. To reply to the note, click 

. The reply box opens in the **Note Properties** dialog box. Type the reply, and then click **OK**.
6. To delete the note, right-click the result and choose **Delete Note**.

To Add or Replace a Tag

1. In the  Interference Detection pane, select the item to add a tag or replace the tag.
2. Right-click and choose **Tag**, and then select a tag from the list, or click **Interference Detection** ▸ **Tag**. The **Tag** list opens.
3. Select a tag from the **Tag** list. The tag appears in the **Tag** column in the  Interference Detection pane.

Viewing Legends on Tags

You can click the arrow in the **Markup** group on the **Interference Detection** tab to view the Tag **Legend**. The legend includes the name and image of each tag and an optional comment.

To Create a Tag for Interference Detection

1. Click **File** ▸ **Creo View Options** . The **Creo View Options** dialog box opens.
2. On the left, click **Tags**. The **Annotations-Tags** options open.
3. Under **Palette Type** select **Multiple** ▸ **Interference Detection**.

   Note

   *You can manage one tag palette for all applications if you choose **Uniform**.*

   The palette opens in the **Tags** box.

4. Click **Add**. The **Edit Tag** dialog box opens.
5. Type a name in the **Tag Name** box and check **Navigator Symbol (16x16)**, and then click **Browse**. The **Select Image** palette opens.
6. To import an image, follow these instructions:
   a. Click **Import**. The **Add Stamp** dialog box opens.
   b. Browse to an image file and click **Open**. The selected image is added to the **Select Image** palette.
7. Select an image from the palette and click **OK**. The image appears alongside **Navigator Symbol (16x16)** in the **Edit Tag** dialog box.
8. Under **Comments** add a description of when to use the tag. The description will appear in the tag **Legend** dialog box.
9. Click **Apply** to apply the options and keep the **Creo View Options** dialog box open, or click **OK** to apply the options and close the dialog box.

To Filter Interference Detection Results

1. Click **Filter** on the Results list in the upper data panel or on the **Full Report**. The **Interference Detection Navigator Filters** dialog box opens.
2. Clear the check box for any instance in the list to exclude it.
3. Click **Apply** to apply the settings, or click **OK** to apply the settings and close the dialog box. The filters are applied either to the Results list or the **Full Report**.

About the Interference View

When you select an interference instance in the Results list, the pair of interfering parts are displayed in the Interference View in the graphics area. You can scroll through the items to view each instance in turn, and you can examine the view with tools that visualize the detected interferences.

When the **Interference View** opens, the noninterference entities in the Model View are grey and transparent by default.

To Explore the Interference View

- To open the **Interference View** in the graphics area, double-click a result in the Interference Detection results on the lower data panel, or click **Interference Detection** ▸  **Open**.
- To display an interference result's compared parts in the **Interference View** display, double-click it.

- To scroll through the results, click **Interference Detection** ▸ **Next** or **Interference Detection** ▸ **Previous**.
- To display all detected interferences, click **Interference Detection** ▸ **Show All**.

To Change the Display of the Model View

When the Interference View is open, you can change the appearance of noninterfering parts in the model view:

- To show only interference entities, click **Interference Detection** ▸ **Display Selected Pairs**.
- To show all parts in the view, click **Interference Detection** ▸ **Display All Inputs**.

Inspecting the Interference View

The Interference View displays the following aspects of the Interference Detection tools:

Tool	Interference Type	Display	Default
Section	Interference, Contact	Section	Shown in the middle of the bounding box.
Bounding Box	Interference, Contact	Location of the interference	**Show Bounding Box Dimensions** is not displayed.
Min. Distance	Clearance	Minimum required clearance distance between two parts	
Volume	Interference	Volume of the penetration	
Contour	Interference	Contour of the penetration	

To Modify Display Colors for Interference Detection

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. In the **Showing** box, select **Active View**.

3. On the left, click **Interference**. The **Model-Interference** options open.
4. Select one of these check boxes and click the color button for the item for any of the following options:
   - **Part 1**
   - **Part 2**
   - **Contour**
   - **Volume**
   - **Bounding Box**
   - **Model Color**

   The **Choose Color** dialog box opens.
5. Select a color from the **Basic colors** list or from the color wheel, or set the **Red**, **Green**, and **Blue** values.
6. Click **OK**.
7. To set a color for another item, repeat steps 4–6.
8. To change the **Transparency Default** of the **Noninterfering Component Display**, type or scroll to the required value.
9. Click **Apply** to apply the options and keep the **Creo View Options** dialog box open, or click **OK** to apply the options and close the dialog box.

Comparing Interference Detection Results

About Comparing Interference Detection Results

There are two ways to compare Interference Detection results:

- **Compare**—Compares the recently performed query with a saved query. The detection settings and the input data can vary.
- **Recalculate**—Compares the results of two calculations of the same input data where changes were made to the location of some parts. Recalculate can also compare calculations if changes were made to some of the test settings, such as changing the detection option from **Interference** to **Clearance**.

To Compare Two Queries

1. Perform an interference detection calculation.
2. Click **Interference Detection** ▸  **Compare**. The **Compare Interference Results** dialog box opens. The **Active Set** contains the present results.
3. Select a **Referenced Set** from the list of previous interference detection calculations.
4. Click **Save as new Report** to save the results as a new results set. Otherwise the results will overwrite those of the active set. The **Report Name** dialog box opens.
5. Type the new name and then click **OK**.
6. Click **Compare**. The new query results appear in the  Interference Detection pane in the lower data panel.

To Recalculate Interference Detection Queries

1. Save an interference detection query as an annotation set.
2. Update the model.
3. Click **Interference Detection** ▸  **Recalculate**. The query results override the previous results in the  Interference Detection pane in the lower data panel.

Understanding the Status of Query Recalculations and Query Comparisons

You can compare the results of two calculations of the same query, or you can compare the results of two different queries. When you compare the results of two calculations of the same query, the status of the item in the  Interference Detection pane is assigned as follows:

- **Resolved**—The interference appears in the first calculation but not in the second.
- **Exists**—The interference appears in both calculations.
- **New**—The interference appears in the second calculation.

When you compare the results of two different queries, the status is assigned as follows

- **Resolved**—The interference appears in the reference set but not in the active set.
- **Exists**—The interference appears in both the reference set and the active set.
- **New**—The interference appears in the active set but not in the reference set.

Note

The first time a query is performed, the resulting status for all interference sets is ***New****.*

Interference Detection Results Reports

About Interference Detection Results Reports

You can generate and export summary reports and full reports:

- The summary report includes general information about the progress of the design procedure. It is useful for managers.
- The full report provides specific details in addition to the general information, allowing hands-on users to read and analyze results case by case.

Both summary and full reports provide this information about the current calculation:

- **General Details**
- **Interferences by Type and Tags**
- **Interferences by Top Level Selection**—Lists the groupings between which interference cases were found and the number of cases between the two groupings.

The **Full Report** also includes information on **Interferences by Components** and items added in **Tag** and **Note**. You can filter the results in the **Interferences by Components** partition. Reports do not include resolved interferences.

Save or export the reports using one of the following file formats:

- CSV
- HTML

To Generate and Export an Interference Detection Results Report

1. Select a report:

- **Interference Detection ▸ Summary Report**
- **Interference Detection ▸ Full Report**

The **Summary Report** or **Full Report** dialog box opens.

2. To export the report, do the following operations:
   a. Click **Export**. The **Export Interference Detection Report** dialog box opens.
   b. Browse to the folder in which to save the report.
   c. Click **Save**.

18

Checking Designs

About Design Check Mode

Use Design Check mode to computerize the design check process. You can tag one or more parts, portions of geometry, and Model Annotations to signify information about the item. You can create and define your own set of tags. The following are examples of information that a tag can convey:

- Status
- Related tasks
- Assignment

After you tag items, you can perform the following operations in Design Check mode:

- Lock the tag
- Add a tag note
- Reply to an existing tag note
- Change the tag
- View a list of the tagged objects
- View a legend for the tags
- Generate a report that lists the history of tag changes for the item
- Save the tags and tag notes as an annotation set

You can configure Design Check mode to make tags recognizable at a glance. Set a neutral color for the display of the structure, geometry, and all of the associated Model Annotations. Assign a color to each tag that you apply to the item. You can define and design any number of other tags for use in Design Check mode. Four tags are included with Creo View. Custom tags and out-of-the-box tags can be modified and deleted.

Access to Design Check functions depends on your edition of Creo View and whether you have a Design Check license installed.

About Design Check Review Mode

You have read-only access to Design Check data in Design Check Review mode. This data includes the following information:

- Tag notes
- Information on the  pane of the upper data panel
- Summary report
- Legend

- Review history
- Tag color in the graphics area

Access to Design Check Functions

Access to the Design Check functions depends on your edition of Creo View and whether you have a Design Check license installed.

Creo View Configuration	Design Check Functions
Creo View with a Design Check license	Read and write
Creo View without a Design Check license	Read-only
Creo View Lite	Read-only
Creo View Express	None

You can also set Creo View to open Design Check information as read-only data.

Working with Tags

About Tagging Designs

When you tag an item in a design, it is added to a list of tagged items in the upper data panel. If the tag includes a color, then the item appears in the graphics window with the color. Tags are only visible in Design Check mode. They do not change the color of the item in the structure. You can tag the following items:

- Part
- Face
- Edge
- Point
- Model Annotation

A Model Annotation is an annotation that is created in the CAD tool along with the structure. You cannot tag a markup created in Creo View. You can assign one tag to a part and a different tag to a piece of the part's geometry.

To Tag an Item

1. On the **Home** tab, in the **Model-Based Design** group, click **Design Check** ▸ **Design Check**. Design Check mode is activated.
2. Select the part, geometry, or Model Annotation that you want to tag.

3. On the **Design Check** tab, in the **Design Check Tags** group, select a tag to apply to it. The item is tagged.

   Note

   You can also use the keyboard shortcuts for Design Check mode to apply tags.

4. To lock the tag, on the **Design Check** tab, in the **General** group, click **Locked**.
5. On the **Design Check** tab, in the **Close** group, click **Close Design Check**.

Keyboard Shortcuts for Design Check Mode

Tags in the **Design Check Tags** group each have a corresponding number based on the order they appear. After you select an item to tag, you can press CTRL+1 to apply the first tag in the **Design Check Tags** group, CTRL+2 to apply the second tag, and so on.

Default Tags for Design Check Mode

The following default tags are included with Creo View:

Symbol	Highlight Name	Description
	Accept	Approves the object.
	Reject	Rejects the object.
	In Work	Flags the object for checking.
	None	Adds the object to the  pane of the upper data panel without assigning a specific status.

Selecting a Tagged Item

Select an item with a tag from one of the following locations:

- Graphics area
- Structure Tree
- Any pane of the upper data panel

To Lock or Unlock a Tag

1. In Design Check mode, select the part, geometry, or Model Annotation whose tag you want to lock or unlock.
2. On the **Design Check** tab, in the **General** group, select or clear the **Locked** check box.

Locking the Tag

Locking prevents an unintentional change to the tag, but it does not limit access to changing the tag. You can lock a tag when you apply it, or you can lock or unlock an existing tag. In Creo View, any user can lock or unlock a tag. When a tag is locked, you cannot do the following operations:

- Change the tag
- Add a note
- Delete the tag

Note

When a part's tag is locked, for the part's geometry you can still add or change a tag.

Lock or unlock a tag from one of the following locations:

- On the **Design Check** tab, in the **General** group
- From the shortcut menu in the  pane of the upper data panel
- In the **Design Check Tag Properties** dialog box

When you define a tag, you can set the locking mechanism to one of the following states:

- **Manual Lock**—The tag must be locked manually.
- **Automatic Lock when created**—The tag is locked automatically.
- **Disabled Lock Mechanism**—The tag cannot be locked.

You can also configure a tag to lock a note when the tag is locked.

To Change a Tag

You can only change a tag that is unlocked.

1. In Design Check mode, select the part, geometry, or Model Annotation whose highlight you want to change. The **Design Check Tags** group becomes available.
2. On the **Design Check** tab, in the **Design Check Tags** group, select a new tag.

Note

You can also change the tag using the upper data panel's shortcut menu, the ***Design Check Tag Properties*** *dialog box, or CTRL+1 for the first tag, CTRL+2 for the second tag, and so on.*

Changing a Tag

Change an item's tag from one of the following locations:

- On the **Design Check** tab, in the **Design Check Tags** group
- From the shortcut menu in the  pane of the upper data panel
- In the **Design Check Tag Properties** dialog box

To Add a Tag Note

1. Make sure that Design Check mode is activated and that at least one part or Model Annotation has a tag.
2. Select the highlight to which you want to add a note.
3. On the **Design Check** tab, in the **Properties** group, click **Tag Note**. The **Design Check Tag Properties** dialog box opens.
4. In the **Notes** box, type a comment.
5. Customize the note with the following settings for the graphics area:
   -  —Notes are expanded.
   -  —Notes are collapsed.
   - **Font** size
   - Text color
   - **Background** color
6. Click **OK**.

To Reply to a Tag Note

1. Double-click the tag note to which you want to reply. The **Design Check Tag Properties** dialog box opens.
2. In the **Notes** box, click **Reply**. A reply box opens.
3. Type a reply and click **OK**.

Selecting a Tag Note

Select a tag note using one of the following methods:

- In the graphics area, double-click the note.
- In the graphics area, select the note, right-click, and choose **Edit Properties**.
- On the  pane of the upper data panel, double-click  for the item.
- Select the note. On the **Annotations** tab, in the **General** group, click **Edit Properties**.

To Delete a Tag

1. Make sure that Design Check mode is activated.
2. Select the item whose tag you want to delete.
3. On the **Design Check** tab, in the **General** group, click **Delete** ▸ **Delete Entire Tag**. The tag, including all properties such as the Review History and notes, is deleted.

To Apply the None Tag and Keep Tag Properties

1. In Design Check mode, select the tag you want to change to **None**.

   Note

   Make sure the tag is unlocked.
2. Select the **None** tag from one of the following locations:

   - On the  pane of the upper data panel, right-click the tag and choose **None**.
   - On the **Design Check** tab, in the **Design Check Tags** group, select **None**.

   The tag is updated and its properties are preserved.

Applying the None Tag and Deleting a Tag

During the design check process, you may want to remove a tag—for instance, to signify the resolution of an issue. To accomplish this, you can apply the None tag, or you can delete the entire tag. The results of each strategy are listed below.

Action	Result
Apply the **None** tag.	In the graphics area, the item is highlighted with the **None** color. The default color is white.
	In the upper data panel, the item remains in the list of tags.

Action	Result
	In the **Design Check Tag Properties** dialog box, the **Review History** is preserved.
	You can lock the **None** tag. Notes that were previously added are preserved and editable. You can add a new note.
Delete the entire tag.	In the graphics area, the item is displayed in the neutral Design Check color that you set in the **Creo View Options** dialog box. Notes are deleted.
	In the upper data panel, the item is removed from the list of tags.
	In the **Design Check Tag Properties** dialog box, the **Review History** is deleted.
	In a report, the item does not appear.

Getting Tag Information

About a Design Check Report

You can generate and export the following types of Design Check report:

- Summary—Provides each tag, the number of items with that tag, and the total number of tagged items.
- Full—Provides each tagged item and the following additional data for each item:
  - Type
  - Name
  - Status
  - Locked or unlocked
  - Note
  - Associated component
  - User that first tagged the item
  - Date and time of the first tag
  - User that last modified the tag
  - Date and time of the last modification

Export the report to one of the following file formats:

- CSV—CSV file for use in a spreadsheet
- Web page—HTM or HTML file for use in a Web page

To Generate and Export a Design Check Report

1. Do one of the following operations, depending on the mode:
   - Design Check mode—On the **Design Check** tab, in the **General** group, click **Report** and select one of the following types:
     - **Summary Report**
     - **Full Report**
   - Design Check Review mode—On the **Design Check** tab, in the **General** group, click **Summary Report**.

   The **Summary Report** or **Full Report** dialog box opens.
2. To export the report, do the following operations:
   a. Click **Export**. The **Export Design Check Report** dialog box opens.
   b. Browse to the folder where you want to save the report.
   c. In the **File name** box, accept the default name or type a new one.
   d. In the **Type** box, select a file format:
      - `CSV`
      - `Web page`
   e. Click **Save**.

To Filter the Design Check Display in the Upper Data Panel

1. Make sure that Design Check mode is activated and that you have at least two tags.
2. In the upper data panel, click  . The Design Check pane opens.
3. On the Design Check pane, click **Filter**. The **Design Check Navigator Filters** dialog box opens.
4. To filter the items by type, in the **Design Check Navigator Filters** dialog box, select one or more check boxes to display the corresponding type of item.
5. To filter the tags, in the **Design Check Navigator Filters**, under **Tag**, select one or more of the check boxes to display the items with the corresponding tag.
6. Click **OK**.

Understanding Symbols on the Upper Data Panel's Design Check Pane

The  pane of the upper data panel contains symbols that indicate the type of item with a tag, whether the tag is locked, and whether a note is attached.

Symbol	Description
	Part
	Face
	Edge
	Point
	Note
	Measurement
	Locked
	Note attached

To View the Review History

1. Select the tag for which you want to see the review history.
2. On the **Design Check** tab, in the **Properties** group, click **Review History**. The **Design Check Tag Properties** dialog box opens with the tag changes listed in the **Review History** area.

To View a Legend for the Tags

1. Make sure that Design Check mode or Design Check Review mode is activated and that you have at least one tag.
2. Do one of the following operations, depending on the mode:
   - Design Check mode—On the **Design Check** tab, in the **Status** group, click the arrow.
   - Design Check Review mode—On the **Design Check** tab, in the **General** group, click  **Legend**.

Creating and Modifying Tags

About Creating and Modifying Tags

You can define any number of tags for use in Design Check and Interference Detection modes in addition to the tags that are included with Creo View. Give users in your organization access to some or all of the tags. When you define a tag, you assign one or more of the following properties to it:

- Color in the graphics area
- Symbol for the Ribbon and the upper data panel
- Information on when to use the tag

You can also edit any existing tag. Manage the tags using the **Creo View Options** dialog box. The properties of the tags are global properties. When you create a tag, you can add comments to a legend. Use the comments section to explain its use.

Users outside your organization can see your tags, and they can change the tag. They cannot apply your tag to the design.

To Create a Tag

1. Click **File** ▸ **Creo View Options** . The **Creo View Options** dialog box opens.
2. On the left, click **Tags**. The **Annotations-Tags** options open.
3. Select a **Palette Type** option:
   - **Uniform**—To manage one palette for all applications.
   - **Multiple** ▸ **Design Check**—To create a tag only for Design Check
   - **Multiple** ▸ **Interference Detection**—To create a tag only for Interference Detection
4. Click **Add**. The **Edit Tag** dialog box opens.
5. Type a name in the **Tag Name** box.
6. To display a tag in a different background color, follow these instructions:
   a. Under **Appearance**, select the **Tag Color** check box.
   b. Click the color button. The **Choose Color** dialog box opens.
   c. Select a color from the **Basic colors** list or from the color wheel, or set the **Red**, **Green**, and **Blue** values.
   d. Click **OK**.
7. Select the **Navigator Symbol (16x16)** or **Ribbon Icon (32x32)** box and click **Browse**. The **Select Image** dialog box opens.

8. To import an image, follow these instructions:
    a. Click **Import**. The **Add Stamp** dialog box opens.
    b. Browse to the image file that you want to add and click **Open**. The selected image is added to the tags display in the **Select Image** dialog box.
9. In the **Edit Tag** dialog box, under **Options**, select or clear the **Show confirmation message when tag is changed** check box.
10. Under **Lock Mechanism**, select a lock setting.
11. To protect against overwriting for locked notes, select the **Switch Note to 'Read Only' when locked** check box.
12. In the **Comments** box, type more information about the tag, or type instructions on when to use the tag.
13. Click **OK**.

To Redefine a Tag

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, click **Design Check**. The **Design Check-Main** options open on the right.
3. Select the tag to redefine, and then click **Edit**. The **Edit Tag** dialog box opens.
4. Change one or more tag settings:
    - Name
    - Color
    - Symbol for the upper data panel, the Ribbon, or both
    - Locking options
    - Comments

To Add Symbols for a Tag

1. In the **Edit Tag** dialog box, select the check box for one or both of the following items:
    - **Navigator Symbol (16x16)**
    - **Ribbon Icon (32x32)**
2. Click **Browse**. The **Select Image** dialog box opens.
3. Select an image from the dialog box and click **OK**, or to import images, follow these instructions:
    a. Click **Import**. The **Add Stamp** dialog box opens.

b. Select the image that you want to add and click **Open**.

c. Repeat steps a-b until all the desired images are in the **Select Image** dialog box.

d. Select an image and click **OK**.

To Set Design Check Options

1. Click **File** ▸ **Creo View Options** . The **Creo View Options** dialog box opens.
2. On the left, under **Model**, click **Design Check**. The **Model-Design Check** options open.
3. Under **Options**, select or clear the **Highlight reference geometry when Model Annotation is selected** check box.
4. To display parts, geometry, Model Annotations, or all three in a neutral color in Design Check mode, do the following operations:

   a. In the **Showing** box, select **Active View**.

   b. On the left, under **Model**, select **Design Check**. The **Model-Design Check** options open.

   c. Under **Initial State**, select or clear the check boxes for either or both of the following options:

      - **Geometry Color**
      - **Model Annotation Color**

   d. Click the color button for the item to define. The **Choose Color** dialog box opens.

   e. Select a color from the **Basic colors** list or from the color wheel, or set the **Red**, **Green**, and **Blue** values.

   f. Click **OK**.

   g. To set a neutral color for the other item, repeat steps d-f.
5. Click **Apply** to apply the options and keep the **Creo View Options** dialog box open, or click **OK** to apply the options and close the dialog box.

19

Adding Construction Geometry

About Construction Geometry

You can use construction geometry tools to create points, planes, curves, axes, coordinate systems, and lines in 3D views. You can create points, axes, and lines in 2D Drawing views. Construction geometry appears in the 2D or 3D view, and is also displayed in the Annotation Sets pane.

You can select construction geometry to perform certain viewing operations, such as translating one or more components along an axis, rotating about an axis, cutting a section plane, or taking measurements. You can load, save, and clear construction geometry you've created.

You can use **Search** to find construction geometry in the structure, and you can use the **Save** button to save the construction geometry.

Create the following types of construction geometry:

- **Point**
- **Plane**
- **Curve**
- **Axis**
- **Coords**—Coordinate system
- **Line**

To Create Construction Geometry

To create construction geometry, use the commands in the **Constructions** group on the **Markup** tab. When you select a construction geometry command, a corresponding collector window displays. Click in the 3D view to select a reference, and use the **Add** button to add rows and click to select new references. When you select an item to populate the collector , the available constraints (drop-down menu) vary based on your selection in the view. The OK button becomes available when a construction geometry can be created. Click OK to create and select the construction geometry.

Construction Geometry References

Use the following references to create construction geometry:

Type	Reference
Point	Vertex Edge Curve

Type	Reference
	Line Axis Plane Coordinate system
Axis	Two points Vertex Construction geometry point Edge, curve, line, or axis Circular edge or curve Surface Plane
Primary axis of geometry	An edge or a curve of an object
Normal vector	Normal vector of the surface at the point you select on the object
Plane	
Coords	Point Cylindrical surface Coordinate system
Curve	Point Surface
Line	Point Curve or edge Circular entity

Using the Bounding Box as Construction Geometry

The bounding box of a component can be designated as construction geometry. You can rotate the object in the bounding box, and you can use the bounding box to view and manipulate the object as you would view and manipulate an axis, point, or plane. When this option is selected, the bounding box always appears around the object as construction geometry, and it does not indicate selection.

Loading Construction Geometry

In the Constructions group in the Markup tab, you can click the Load command to display the **Load Construction Geometry** dialog box, where you can browse to select the geometry you want to import.

20

Working with Groups

About Groups

You can create groups to manage and maintain a set of entities that share a common purpose or meaning. Create groups by selecting MCAD or ECAD entities in the graphics area, from the structure, or by redirecting search results. The group functionality is context-sensitive. MCAD group options open to MCAD data, and the ECAD group options open to ECAD viewables.

Perform the following operations on groups:

- Name
- Edit the name, content, and display
- Sort the contents
- Add comments

You can use groups to modify the view in the graphics area. Select a group, and then select a view operation to perform on the group members:

- Isolate
- Hide and unhide
- Zoom
- Highlight ECAD groups

Groups are on the  Groups pane of the upper data panel. When you select a group name from the list, the members appear in the MCAD **Part Name** partition, or the ECAD **Group Contents** partition below. You can create an empty group and add members later.

You can save the contents of selected groups, or of all groups.

In Creo View, you can create and modify groups. In Creo View Lite, you can view groups.

To Save One or More Groups

1. On the  Groups pane, select the groups to save.
2. Perform one of the following actions:
   - Right-click and choose **Save** from the shortcut menu.
   - Click **Groups** ▸  **Save** ▸ **Save Group**.

To Save All Groups

1. In the upper data panel, click  . The Groups pane opens.
2. Perform one of the following actions:
   - Right-click in the  Groups pane and choose **Save All** from the shortcut menu.
   - Click **Groups** ▸  **Save** ▸ **Save All Groups**.

Creating MCAD Groups

About MCAD Groups

You can select parts and subassemblies for four types of groups:

-  **Static**—Groups parts and assemblies that you select.
-  **Nested**—Creates a group of groups. It can contain static, nested, search and rule groups. The  Nested group appears in the Groups pane with its child groups.

  When you select a nested group, all the members of all of its children appear under **Part Name**. When you select a child group, only its members appear in the **Part Name** partition.

  You can delete only entire child groups from a Nested group. You cannot delete parts individually.
-  **Search**—Saves the results of a search as a group in the **Part Name** partition of the Groups pane instead of in the **Part Name** partition of the search results pane.
-  **Dynamic**—Defines criteria to subdivide MCAD groups according to the hierarchy of parts or the value of a specific attribute. For instance, a group called Motor_Parts subdivided according to the attribute material, will have subgroups called Motor_Parts_steel and Motor_Parts_aluminum.

  Create a Dynamic group without parts, import the parts from an existing MCAD group, and then select the criteria for inclusion into the subgroups.

You can share an MCAD group. Shared groups are applied to any MCAD representation that you open. When a group is shared,  appears alongside the group icon in the  Groups pane.

To Create a Static Group

You can create a Static group with or without preselecting entities.

1. To include selected entities, select one or more entities from the  View pane, the  ECAD Results pane, the **Part Name** partition of the Groups pane, the search results list, or the graphics area, and then click **Create Static Group** or **Create Group** ▸ **Static** in one of these locations:
   - **Groups** tab
   - **Structure** tab
   - **Model Parts** tab

   The **Group Name** dialog box opens.
2. Type a name for the group and click **OK**.

To Create a Nested Group

You can create a nested group in two ways:

From the Groups Pane

1. Select groups in the  Groups pane, and then click **Create Group** ▸ **Nested**. The **Group Name** dialog box opens.
2. Type a name for the group and click **OK**.

or

From the Group Name Dialog Box

1. Click **Create Group** ▸ **Nested**. The **Group Name** dialog box opens.
2. Type a name for the group click **OK**. The group is created.
3. In the  Groups pane, drag group names to the new group.

To Create a Search Group

You can create a search group in two ways:

From the Group Name Dialog Box

1. In an open search configuration, click **Save As Group**. The **Group Name** dialog box opens.
2. Type a name for the group click **OK**. The group is created.

or

From the Create Group Dialog Box

1. Click **Create Group** ▸ **Search**. The **Create Group** dialog box opens.
2. Type a name for the group and click **OK**. The **Search Product Structure** dialog box opens.
3. Define the search and click **Apply**. The group is created.

To Edit a Search Group

1. Right-click a  Search group and choose **Edit Properties**, or select a  Search group in the  Groups pane and then click **Groups** ▸  **Edit Properties**. The **Properties** dialog box opens.
2. Click **Query**. The **Search Product Structure** dialog box opens.
3. Modify the settings in the **Search Product Structure** dialog box, and then click **Apply**. The group is edited.
4. Click **OK**.

To Create a Rule Group

1. Select a group in the  Groups pane, and then click **Create Group** ▸ **Rule**, or click **Groups** ▸ **Create Group** ▸ **Rule** on the ribbon. The **Group Name** dialog box opens.
2. Type a name for the Rule group and click **OK**. The  **Rule** group is created. By default each group member is a  Subgroup.
3. Right-click the Dynamic group on the  Groups pane and choose **Edit Properties** from the shortcut menu, or select the Dynamic group and click **Groups** ▸  **Edit Properties**. The **Properties** dialog box opens.
4. Under **Source**, select a **Group** from the list.
5. Under **Grouping Criteria**, select **By attribute** or **By hierarchy**, and then select an option from the list.
6. Set the **Group naming criteria**.
7. Select **Shared** to apply the group to other representations.
8. Click **Close**. The  Subgroups update. Select a  Subgroup to view its members in the **Part Name** partition.

Setting Subgroup Criteria

Subgroup criteria divides the group's members into subgroups. There are two categories of subgroup:

- **By attribute**—Includes parts with the selected attribute. The options include attributes from all components in the design.
- **By hierarchy**—Sets a level of hierarchy to appear in the subgroup.
  - **Top level**—Includes only the top level of each entity.
  - **Children**—Includes only the children of entities.

  Select **Include descendants in group contents** to include the children of parts that appear in a subgroup.

Specify the naming of the resulting subgroups under **Group naming criteria**. You can edit the field to change the naming criteria. Use the **Insert Special** list to include an option from the list.

To Share a Group

1. Right-click the group and choose **Edit Properties**, or select an MCAD group in the  Groups pane and then click **Groups** ▸  **Edit Properties**. The **Properties** dialog box opens.
2. Click **Shared**, and click **Close**.

Creating ECAD Groups

About ECAD Groups

You can select objects for an ECAD group from the graphics area, the Viewables pane, search results, or the contents of another group. ECAD groups are viewable-specific. If you have two views of the same ECAD design and add a group in one of them, the new group appears in the other view when it is activated. You can duplicate an ECAD group and save it with another name.

You cannot include markup objects in an ECAD group.

Control the color and visibility of ECAD groups in the **Groups** branch of the **Color and Visibility Controls** pane.

To Create an ECAD Group

You can create an ECAD group with or without preselecting objects.

1. To include selected object, select one or more objects from the  View pane, the graphics area, the  ECAD Results pane, or the **Group Contents** partition of the  Groups pane.
2. Right-click the primary panel and choose **Create Group**, or click **Groups** ▸  **Create Group**.

   The **Create Group** dialog box opens.
3. Type a name for the group and click **OK**. The new group name is added in the  Groups pane in the upper data panel.

To Duplicate an ECAD Group

1. Select a group from the  Groups pane then click **Groups** ▸  **Create Group**. The **Create Group** dialog box opens.
2. Type a name for the new group and click **OK**. The new group is created.

Modifying Groups

About Modifying Groups

You can modify groups in any of these ways:

- Refresh contents
- Add or remove members
- Rename
- Delete
- Add comments

When you open an MCAD group after the model has been modified, use the **Refresh Contents** command to update the group contents. Refreshing a group contents includes or excludes relevant parts in the group.

To Refresh MCAD Group Contents

1. On the  Groups pane, right-click and choose a group.
2. Right-click in the **Part Name** pane and choose **Refresh Contents**. The group contents update.

To Add Entities to a Static or an ECAD Group

You can add entities to a group in different ways:

1. Select one or more entities from the primary panel, the graphics area, or the search results list on the upper data panel.
2. Click  **Add to Group**, and then select the group name from the  list. The entities are added to the group.

or

1. Select the group name from the  group list.
2. Select one or more entities on the primary panel and drag them to the **Part Name** pane.

or

1. Select one or more entities and then right-click and choose **Add to Group**.
2. Select the group name from the list on the shortcut menu.

To Rename a Group

1. On the  Groups pane, right-click a group and choose **Rename** from the shortcut menu, or select the group and click **Groups** ▸  **Rename Group**. The **Group Name** dialog box opens.
2. Type a new name and click **OK**.

To Add Comments to a Group

1. Select an MCAD group in the  Groups pane and then click **Groups** ▸  **Edit Properties**, or right-click the group and choose **Edit Properties**. The **Properties** dialog box opens.
2. Click **Comments**. The **Comments** tab opens.
3. Type your comments and click **Close**.
4. Repeat this procedure to view comments.

To Remove Parts from a Group

1. On the  Groups pane, select the group.
2. Under ECAD **Group Contents** or MCAD **Part Name**, select one or more parts, and perform one of these actions:
   - Right-click and choose **Remove from Group** from the shortcut menu.
   - Click **Group Contents** ▸  **Remove from Group**.

To Delete a Group

1. On the  Groups pane, select the group, and perform one of these actions:
   - Right-click and choose **Delete** from the shortcut menu.
   - Click **Groups** ▸ **Delete Group**.

Modifying Views Using Groups

About Modifying Group Views

You can modify group views in any of these ways:

- Show or hide group members' paths
- Filter the list of groups
- Filter the list of members of one group
- Isolate, show, or hide groups
- Highlight groups
- Zoom

About Showing or Hiding Group Paths

Use the  **Show Paths** option to show or hide the structure paths to MCAD group members in the **Part Name** partition. The option that you choose will apply to all groups that you select.

To Show or Hide Paths for Part Names

1. Select a group in the  Groups pane.
2. Under **Part Name**, select any part, and then click **Group Contents** ▸  **Show Paths**, or right-click **Part Name** and choose **Show Paths**. The full path of each part is shown or hidden.

To Modify the View Using Groups

In the  Groups pane, select the group and then choose a command from the ribbon, or right-click and choose it from the shortcut menu. The view is updated.

- **Open in new view** (MCAD only)
-  **Isolate Group**
-  **Hide**
-  **Unhide All**
-  **Zoom to Group**
-  **Highlight Group** (ECAD only)

About Filtering the List of Groups

You can filter MCAD groups by type. You can also apply a filter to show shared groups or saved groups in the current structure. Only the selected group types are displayed. You can filter the display of one or more ECAD and MCAD groups by name.

To Filter the List of Groups

1. In the  Groups pane, right-click and choose **Filter Group List**. The **Filter Groups** dialog box opens.
2. To filter by name, select **Names matching** and then type the name of a group in the **Enter text to match** box.
3. To filter by local or shared status, select **Only groups stored** and then select either **In the current structure** or **Shared across all structures**.
4. To filter by type in an MCAD project, select **Groups of type**, and then select one or more group types to display.
5. Click **Make Default** to apply the filter settings to all groups.

6. Click **OK**. The list is filtered.
7. To cancel the filter settings, in the  Groups pane, right-click and choose **Reset Filter**.

To Filter the List of Items in One Group

1. Under ECAD **Group Contents** or MCAD **Part Name**, right-click and choose **Filter Group Contents**. The **Filter Group Contents** dialog box opens.
2. Select **Names matching** and then type the name of a group to display in the **Enter text to match** box.
3. Click **Make Default** to save the settings as the default.
4. Click **OK**. The Groups list is filtered.
5. To cancel the filter settings, in the ECAD **Group Contents** or MCAD **Part Name** partition, right-click and choose **Reset Filter**.

21

Watermarks

About Watermarks

Watermarks are text, images, or both that are overlaid on a view when it is displayed or printed. This information can be derived from any of the following sources:

- System attributes such as time and date
- PDM such as release status
- User-defined attributes such as company name and logo

You can specify a number of different watermarks to use, based on values of attributes in the product structure. The **Property Name** field defines the name of the property to look for, and the table lists the possible values for this property, along with the watermark to use when the specified attribute has a matching value in the product structure. If the property name or value is not found in a particular file, the **Default Watermark** is used instead.

To Define and Apply Watermarks

1. Click **File** ▸ **Creo View Options**. The **Creo View Options** dialog box opens.
2. On the left, under **General**, click **Watermarks**. The **General** — **Watermarks** dialog box opens.
3. To add a watermark perform theses steps:
   a. Click 

. The **Add Watermark** dialog box opens
   b. Type a name and browse to and add an `.ini` file, and then click **OK**.
4. To set a watermark as default, select the watermark and click **Set As Default**.
5. To remove a watermark, select the watermark and click **Delete**.
6. Select or clear the **Include watermark when printing** box.
7. Select or clear the **Include watermark when viewing** box.
8. Click **Apply** to apply the options and keep the **General** — **Watermarks** dialog box open, or click **OK** to apply the options and close the dialog box.

22

Saving, Printing, and Sending Files

About Saving MCAD and ECAD Files

You can save MCAD structures, ECAD designs, and their associated viewables and annotation sets:

- PVZ—Includes files for all viewables and annotations.
- PVS—Saves the information as a directory and references the viewable and annotations files.

To Save an MCAD and ECAD File

Click **File** ▸ **Save All**. The file is saved.

About Printing Viewables

You can print one or more parts or viewables associated with a structure at one time. You can print multiple items in these ways:

- List—Each selected item is a separate print job. You can select **Print Settings** for each item.
- Combined—All selected items are included in one print job. You can select one set of **Print Settings** for all items in the print job.

You can include Illustration or Document views in the list of a multiple print job, but you cannot include them in a combined print job, or as part of a single job derived from a multiple print job. Use the advanced options to print a background color and control the margins of printed images.

To Set Advanced Print Options

1. Access the advanced print options in one of these ways:
   - From the **Print** dialog box—Click **Preferences** ▸ **Advanced**.
   - From the **Print Multiple Items** dialog box—Click  **Print Settings** ▸ **Global Advanced options**.

   The **Advanced Print** dialog box opens.
2. Select one or more of these options for your viewable:
   - **3D Model**—**Print background color**
   - **Drawing**—**Print background color**

- **Drawing**—**Monochrome printing**
- **Image**—**Print without margins**

3. Click **OK**. The **Advanced Print** dialog box closes.

To Print a Viewable

1. Make sure the viewable you want to print is open and that the correct view is displayed.
2. Click **File** ▸ **Print**. The **Print** dialog box opens.
3. Under **Printer**, do one or more of the following operations:
   - Select a printer.
   - Select the **Print to file** check box.
   - Click **Properties** and set printer properties.
4. Under **Print Range**, set the level of **Zoom** for the printout:
   - **As Displayed**—Prints according to the level of zoom you set in the graphics area.
   - **All**—Prints the entire item.
5. Under **Copies**, set the number of copies and select or clear the **Collate** check box.
6. Under **Layout**, set one or more of the following parameters:
   - **Fit to Page**
   - **Scale**—Select a scale from `10` percent to `100` percent.
   - **Rotate and Center for best fit**—The viewable is manipulated to fit the page.
7. Click **Preferences** to set the following options:
   - **Watermark**
   - **Advanced**

   Click **OK** in the **Advanced Print** dialog box.
8. Click **OK**.

To Print Multiple Items

1. Click **File** ▸  **Print** ▸  **Print Multiple Items**, and then click **OK**. The **Print Multiple Items** dialog box opens.
2. Drag the items to the print list from any of these locations:

- Viewables pane of the primary panel
- Annotation sets
- **Views** group of the **Home** tab

3. To print all the viewables associated with a structure, including annotation sets, click **Add All Viewables**.
4. To reorder the printing list, click  ,  ,  , and  .
5. To remove an item from the list, click  .
6. To print all the items as a combined print job, click the arrow next to **Print Settings**, and then click **Print as single job**.

   Note

   PDF and ISO files cannot be combined in a single job.
7. To configure the print settings for an item or a combined job, select it and then click **Print Settings**.
8. Click **Print**.

About Print Preview

Use Print Preview to change how the display content appears on the screen and is printed. You can change the page setup, the print settings and scale, and determine whether to include a watermark or not. The status bar provides information regarding the current page number and number of pages or sheets, the paper size, and the name of the currently selected printer.

All view types are supported, except for PDF and IsoDraw . Print preview for pdf is handled in the acrobat window, and Isodraw displays its own print dialog.

Print Preview is available in both Creo View and Creo View Lite

To Preview a Print Job

1. Click **File** ▸ **Print** ▸  **Print Preview**. The Print Preview dialog box opens.
2. Adjust the page settings:
   - **Orientation**—Switches the pages between portrait and landscape layouts.
   - **Size**—Selects the page size for the current view.
3. Select one or more of these options:
   - **Print Watermarks**—Includes watermark when printing.
   - **Print background color**

- **Print without margins**
- **Print monochrome**
- **Print Watermarks** ▸ **Edit Watermarks** to open the watermarks dialog box to select a watermark.

4. Use one of these modes to define the image size and position on the page:
   - **Zoom**
   - **Fit to Page**—Opens scale size options
5. Click **Options** to open the **Document Properties** dialog box.
6. Click **Print** to open the **Print** dialog box and print the image, or **Close Print Preview** to return.

To Add a Watermark to a Print Job

1. Open the **General-Watermarks** dialog box in one of these ways:
   - From the **Print** dialog box—Click **Preferences** ▸ **Watermarks**
   - From the **Print Multiple Items** dialog box—Click **Print Settings** ▸ **Watermarks**.
   - Click **File** ▸ **Creo View Options**.
2. Under **Default Watermark**, select the watermark to apply.
3. Select the **Include watermark when printing** check box, and then click **OK**.

To Print a PDF

1. Open a PDF in Creo View.
2. Click **File** ▸ **Print** ▸ **Print**. The **Print** dialog box opens.
3. Configure the printing settings and click **OK**. The document is printed.

To Save a PDF Locally

1. Open a PDF in Creo View.
2. Click **File** ▸ **Save Current View As** ▸ **As a PDF file**. The **Save View as PDF** dialog box opens.
3. Select a location and type a name for the file. Click **OK**. A copy of the file is saved.

To Send a View by E-mail

1. Make sure the representation you want to send is open and the correct view is displayed.
2. Click **File** ▸ **Send** ▸ **Active view by E-Mail**. The **Save View As Image-Settings** dialog box opens.
3. Under **Units**, select the unit of measurement for the image:
   - **Inches**
   - **Centimeters**
   - **Pixels**
4. Under **Width** and **Height**, set the dimensions for the image.

   Note

   *When the **Lock aspect ratio** check box is selected, the proportions of the image are preserved. You can change one dimension and the other is updated automatically.*
5. To set a resolution for images measured in inches or centimeters, in the **Resolution** box, type a number of pixels per unit.
6. Click **OK**. An e-mail message opens with the image file attached.

To Configure Print Settings

1. In the **Print Multiple Items** dialog box, select the a viewable, and then click **Print Settings**. The **Print** dialog box opens.
2. Under **Printer**, do one or more of the following operations:
   - Select a printer.
   - Select the **Print to file** check box.
   - Click **Properties** and set printer properties.
3. Under **Print Range**, select one of these options:
   - **Sheets**—Prints selected pages.
   - **All Pages**—Prints the entire item.
   - **Current Page**—Prints current page.

   If enabled, you can set the level of **Zoom** for the printout.
4. Under **Copies**, set the number of copies and select or clear the **Collate** check box.
5. Under **Layout**, set one or more of the following parameters:

- **Fit to Page**
- **Scale**—Select a scale from 10 percent to 100 percent.
- **Rotate and Center for best fit**—Manipulates the item to fit the page.

Note

***Zoom**, **Collate**, **Scale**, **Rotate and Center for best fit** are not available for PDF documents.*

6. Click **OK**. The **Print** dialog box closes.